Magnifi® 3.3

eddyfi™

User's Guide

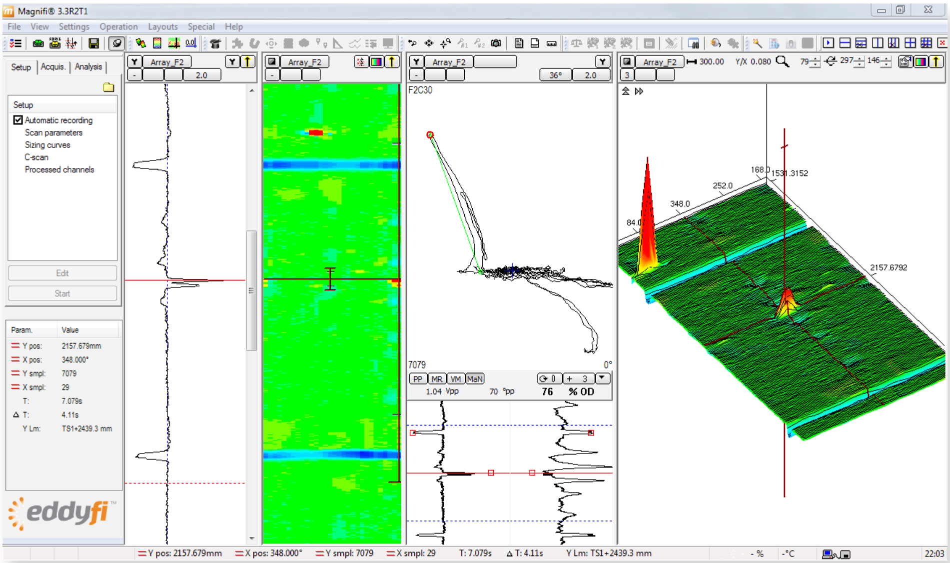


Table of Contents

Introducing Magnifi

1

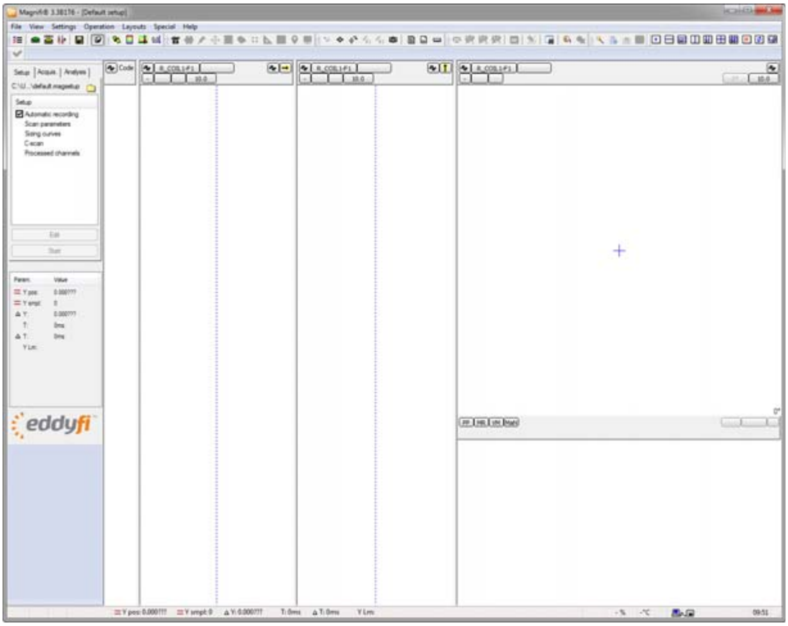
Starting Magnifi

Once you have finished installing or updating the Magnifi software on your workstation (as per the installation booklet that came in the program's box, or from the procedure found in "Reinstalling Magnifi" on page 297), the program's icon appears on your desktop.

Figure 1-1 The Magnifi icon

- To start the program, double-click the program's icon, or select the icon and press ENTER on your keyboard. The program starts and the main window appears.

Figure 1-2 The Magnifi main window



Note *The first time that Magnifi is started on a newly installed workstation, you will be asked to choose a default measurement mode (ASME or EDF). For more information, see "Setting Measurement Conventions" on page 72.*

Understanding the User Interface

Magnifi offers an improved, more modern and effective display mode to interact with the user. Keyboard shortcuts and mouse actions are more in-line with the now-usual Microsoft Windows standards.

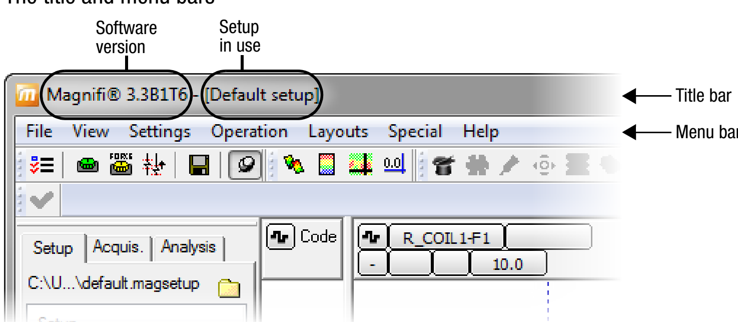
In the following pages, the basic elements of the Magnifi main window will be discussed:

- Title and menu bars
- Status bar
- Toolbars
- Info and control bars
- Data display area

Title and Menu Bars

At the top of the window, you find the title bar and menu bar. The title bar informs you of the software version that you are running and the setup in use.

Figure 1-3 The title and menu bars



Toolbars

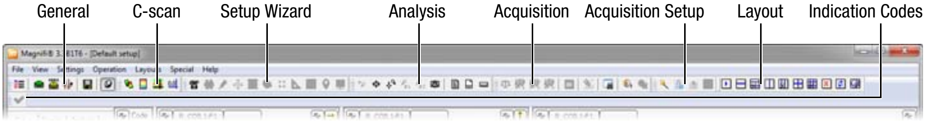
Underneath the menu bar is the default toolbar area, where toolbars appear when you select them on the **View** menu.

The available toolbars are:

- General
- C-scan
- Setup Wizard
- Analysis
- Acquisition Setup
- Acquisition
- Layout
- Indication codes
- Acquisition Notepad (see “Control and Info Bars” on page 5)

Functions performed from these toolbars are explained in context throughout this documentation.

Figure 1-4 The toolbars



Another toolbar that is available from the **View** menu is the **Acquisition Notepad**. By default, this toolbar does not appear in the toolbar area and is explained in more detail in the next section.

Status Bar

The status bar is found in the lower part of the display. It gives most of the information given in the Info bar (see “Control and Info Bars” on page 5), but it also indicates the connection status between the program and an acquisition instrument, the Ectane battery status, and the time of day (synchronized with the workstation internal clock).

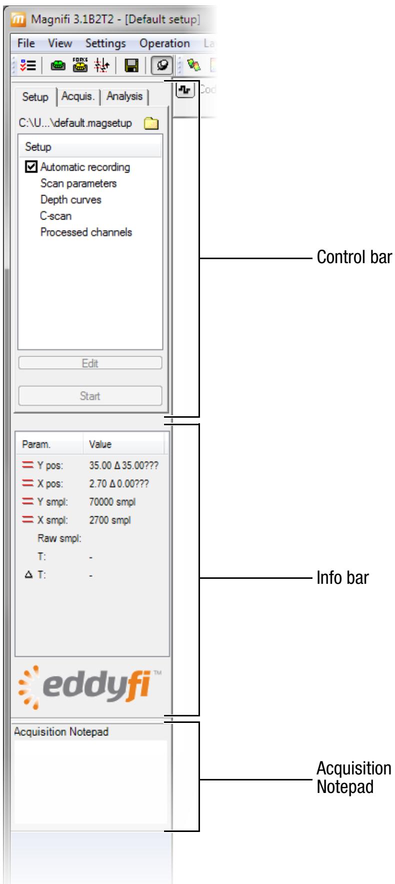
Battery and connection status icons are explained in more details on page 219.

Figure 1-5 The status bar

Control and Info Bars

On the left side of the display, you find the control and info bars, as well as the **Acquisition Notepad** toolbar. These interface elements can be displayed/hidden when selected from the **View** menu.

Figure 1-6 The control and info bars, as well as the **Acquisition Notepad** toolbar



Functions performed from these bars will be explained in context throughout this documentation.

Most of the information given in the info bar also appears in the status bar at the bottom of the display (see “Configuring the Info and Status Bars” on page 7.)

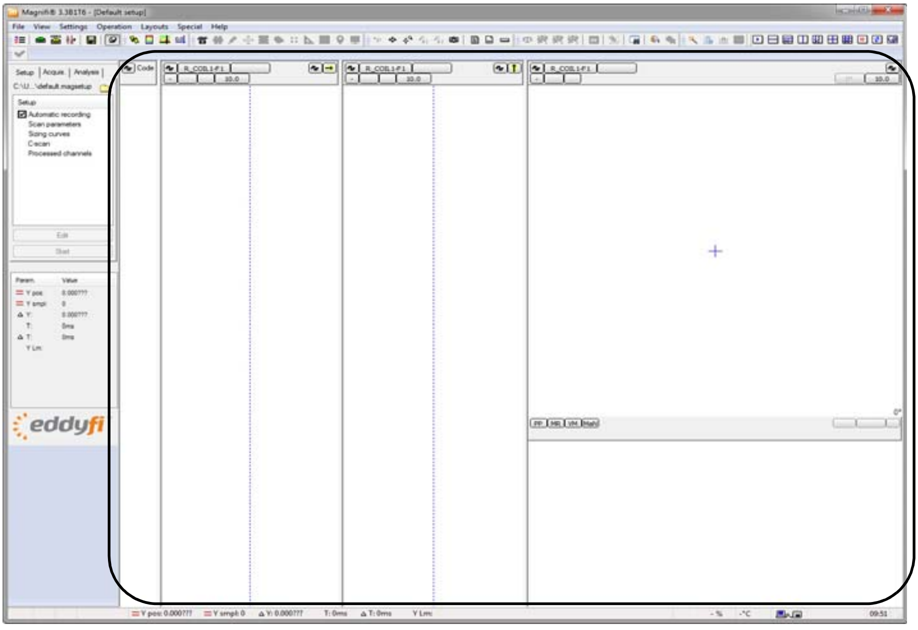
Data Display Area

The display area is where the actual data can be displayed, in various formats:

- Strip charts
- Lissajous
- A-scans
- C-scans
- Voltage planes
- Polar views
- etc.

Important The display area is divided in sections called *panes.* The data displayed in a pane is called a *view.*

Figure 1-7 The data display area



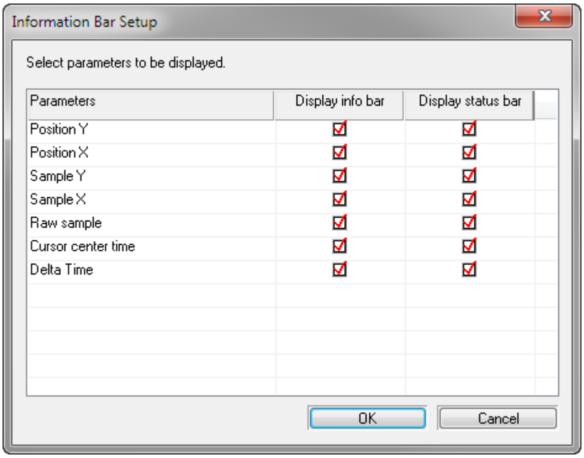
Configuring the Info and Status Bars

You can choose the information that will be displayed in the info and status bars. By default, this information is identical.

To configure the information displayed in the info and status bar:

1. Double-click anywhere in the list found in the info bar. The **Information Bar Setup** window appears. By default, all boxes are checked.

Figure 1-8 The **Information Bar Setup** window



2. Click the check boxes to select (display) or deselect (hide) the information fields to display in the info bar and the status bar.
3. Click **OK** to save changes and return to the Magnifi main window, or **Cancel** to discard the changes that you made before returning to the Magnifi main window.

The info and status bars now reflect the changes that you made.

Note *The parameters available for display depend on the type of scan selected, whether or not encoders are used, etc. For more information on scan type selection, see “Configuring Scan Parameters” on page 104, and for encoder selection and configuration, see “Configuring Ectane ECT Acquisition Setups” on page 81.*

Switching between Operation Modes

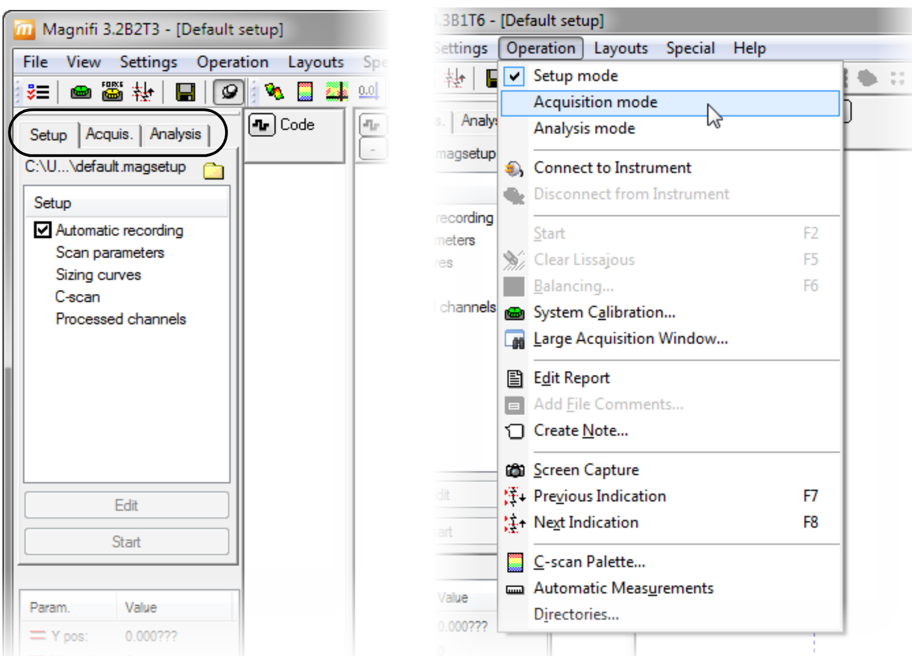
Magnifi offers three operation modes:

- Setup
- Acquisition
- Analysis

There are two different methods of switching between operation modes: from the control bar, or from the **Operation** menu.

- To switch modes from the control bar, click the appropriate tab:

Figure 1-9 Switching operation mode from the control bar (left) and from the **Operation** menu (right)



Managing Toolbars

Toolbars give you a quicker way to access frequently used functions. The preferred position for these toolbars is in the toolbar area, but you can move them anywhere you want on-screen *for the current session.*

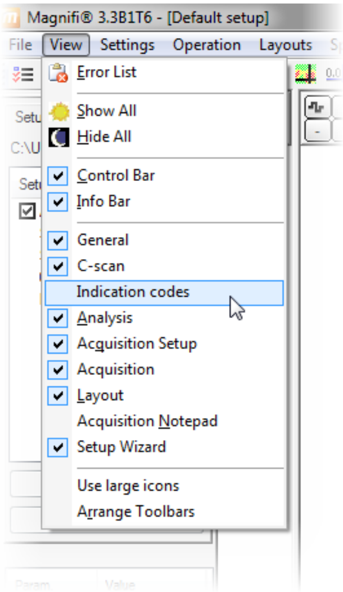
Note *Toolbar position is not saved in the setup file or when you quit the application.*

Displaying/Hiding Toolbars

Toolbars can be displayed or hidden individually or all at once.

- To display or hide all toolbars at once, select **Show All** or **Hide All** from the **View** menu.

Figure 1-10 The **View** menu



- To display or hide a specific toolbar, select it from the **View** menu. Displayed toolbars appear with a check mark next to their name.

Changing Icon Size

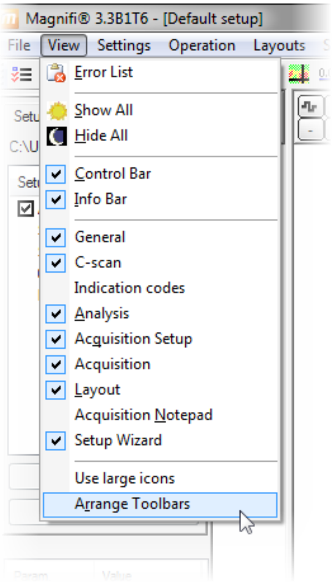
Should you feel that icons are too small for your liking, Magnifi offers the possibility to increase the icon size.

To make icons larger, select **Use large icons** from the **View** menu (see Figure 1-10). Changes will take effect the next time that you start Magnifi.

Rearranging Toolbars

Toolbars can be moved anywhere on screen for the on-going session. To quickly place them back in the toolbar area, in their default location, select **Arrange Toolbars** from the **View** menu.

Figure 1-11 Selecting **Arrange Toolbars**



Managing Layouts

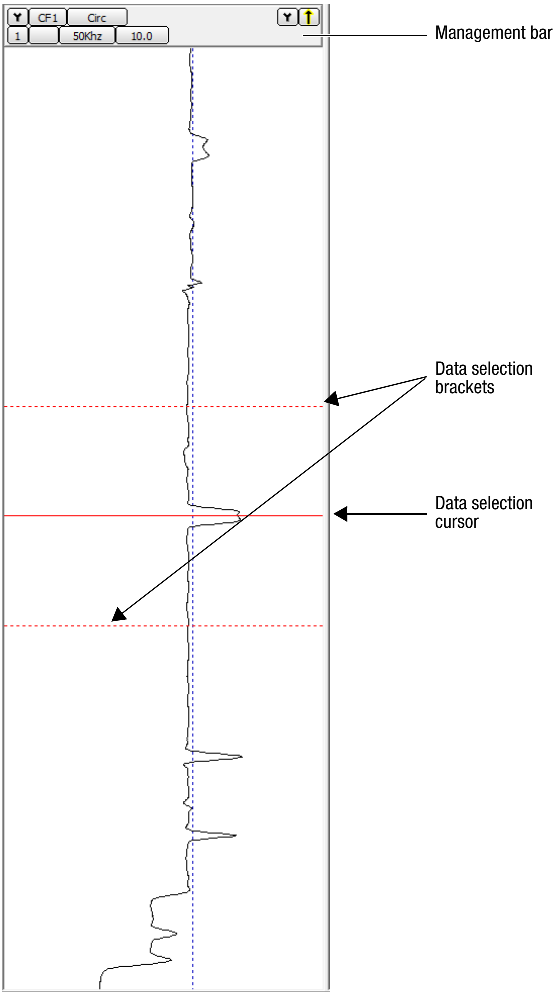
Layouts are a very important part of effective acquisition and analysis tasks. They allow you to view live or prerecorded data in a manner that is more efficient to you. By organizing the various charts and graphical elements available in Magnifi in display elements called *panes,* you can create the layout that will allow you to perform better.

Understanding Basic Layout Elements

Magnifi offers several standard layout elements (strip charts, Lissajous, voltage planes, side views, C-scans [regular and 3D] and polar views [regular and 3D]). These elements are presented in the following pages.

Strip Charts

Figure 1-12 Strip chart



Elements of a Strip Chart

All strip charts are presented the same way, with the same elements, as displayed below:

Figure 1-13 The strip chart management bar

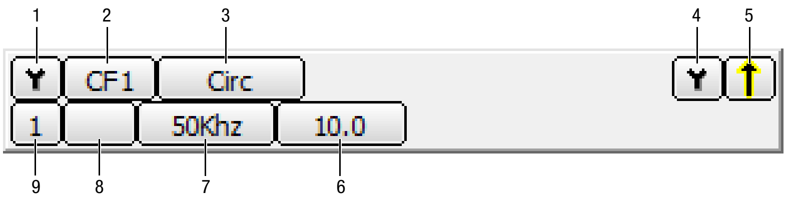
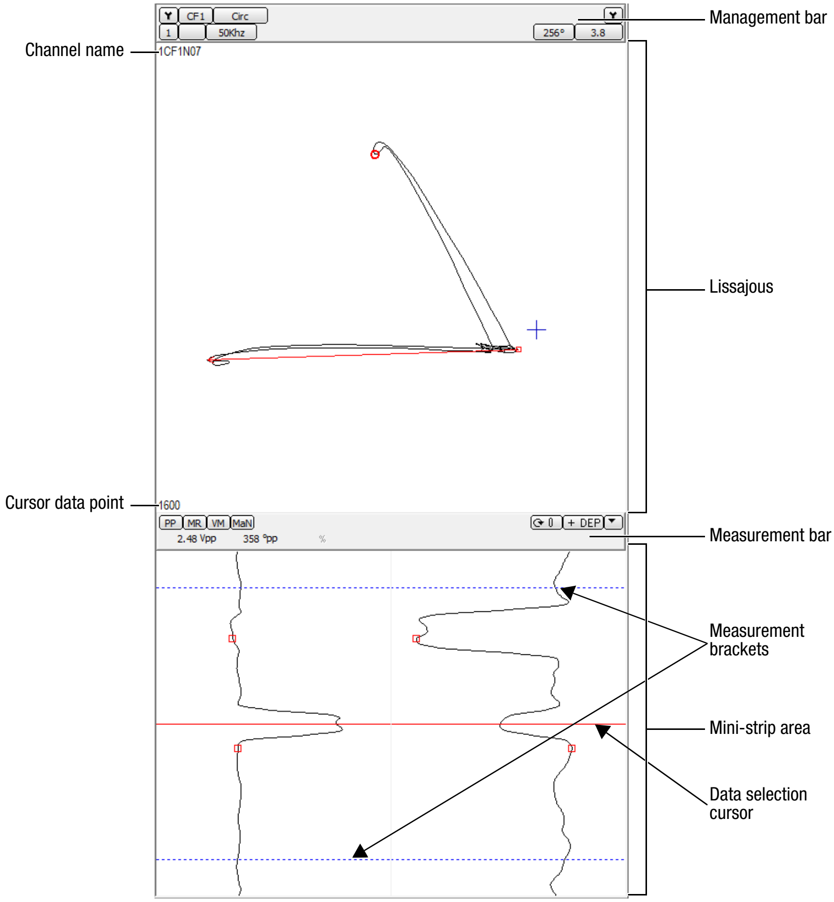


Table 1-1 Description of strip chart management bar items

	Item name	Description
1	Channel/ C-scan list	Gives you access to the list of available channels or C-scans. For more information, see “Setting Displayed Channel/C-scan Information” on page 78.
2	Channel/ C-scan name	Indicates the name of the channel or C-scan. For more information on naming channels, see “Setting Displayed Channel/C-scan Information” on page 78.
3	Channel/ C-scan label	Indicates the label given to a channel. For more information on labeling channels, see “Setting Displayed Channel/C-scan Information” on page 78.
4	Data type	Indicates the type of data displayed (C-scan X or Y component, or channels).
5	Axis	Indicates whether the vertical or horizontal component of the data is displayed.
6	Scale	Indicates the scale of the displayed data. For more information on modifying the scale, see “Measuring Indications” on page 261.
7	Frequency	Indicates the frequency of the displayed data. For more information on frequencies, see “Setting Displayed Channel/C-scan Information” on page 78.
8	Coil	Indicates the coil to which belongs the displayed data. For more information on coil numbers, see “Setting Displayed Channel/C-scan Information” on page 78.
9	Link	Allows you to link views. For more information, see “Linking Views” on page 61.

Lissajous

Figure 1-14 Lissajous



Elements of a Lissajous

All Lissajous are presented the same way, with the same elements, as displayed below:

Figure 1-15 The Lissajous management bar

Table 1-2 Description of Lissajous management bar items

	Item name	Description
1	Channel/ C-scan list	Gives you access to the list of available channels or C-scans. For more information, see “Setting Displayed Channel/C-scan Information” on page 78.
2	Channel/ C-scan name	Indicates the name of the channel or C-scan. For more information on naming channels, see “Setting Displayed Channel/C-scan Information” on page 78.
3	Channel/ C-scan label	Indicates the label given to a channel. For more information on labeling channels, see “Setting Displayed Channel/C-scan Information” on page 78.
4	Data type	Indicates the type of data displayed (C-scan X or Y component, or channels).
5	Scale	Indicates the scale of the displayed data. For more information on modifying the scale, see “Measuring Indications” on page 261.
6	Rotation	Allows you to rotate a signal, and indicates the angle of rotation. For more information on rotating signals, see “Calibrating Systems” on page 238.
7	Frequency	Indicates the frequency of the displayed data. For more information on frequencies, see “Setting Displayed Channel/C-scan Information” on page 78.
8	Coil	Indicates the coil to which belongs the displayed data. For more information on coil numbers, see “Setting Displayed Channel/C-scan Information” on page 78.
9	Link	Allows you to link views. For more information, see “Linking Views” on page 61.

Figure 1-16 The Lissajous measurement bar

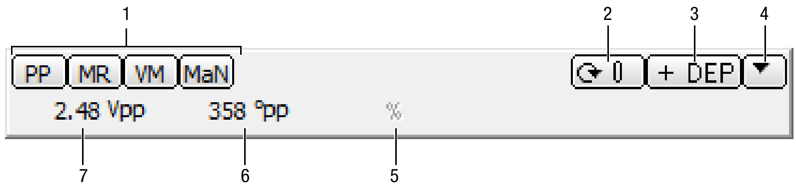
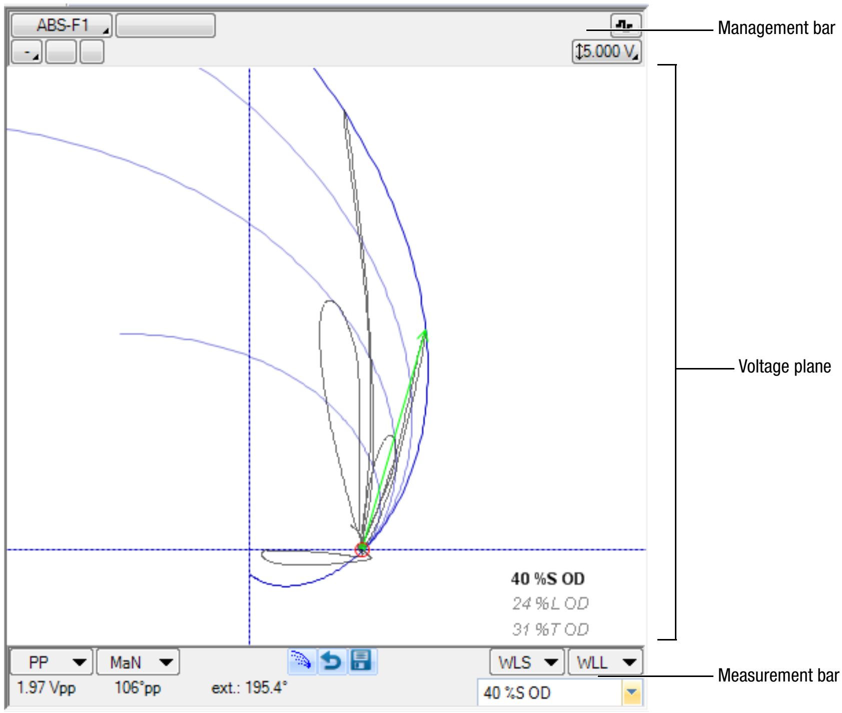


Table 1-3 Description of Lissajous measurement bar items

	Item name	Description
1	Measurement method selection	Allows you to choose a different measurement method for the phase data displayed in the Lissajous. **PP:** peak-to-peak **MR:** maximum rate **VM:** vertical maximum **MaN:** manual For more information on measurement methods, see “Setting Amplitude/Phase Measurement Methods” on page 74.
2	Rotation	Allows you to quickly rotate signals 180°. For more information on rotating signals, see “Calibrating Systems” on page 238.
3	Indication code assignment	Indicates the code that will be assigned to an indication when you click it.
4	Indication code selection	Opens a drop-down list of available indication codes. For more information on indication codes, see “Managing Indication Codes” on page 197.
5	Size	Indicates the size of a signal based on a sizing curve. A value is indicated only if a sizing curve is defined for the displayed channel or C-scan.
6	Measured angle	Indicates the measured angle of the signal inside the measurement brackets. The default measurement mode is defined in the preferences (see “Setting Amplitude/Phase Measurement Methods” on page 74).
7	Measured amplitude	Indicates the measured amplitude of the signal inside the measurement brackets. The default measurement mode is defined in the preferences (see “Setting Amplitude/Phase Measurement Methods” on page 74).

Voltage Plane View

Figure 1-17 Voltage plane view



Elements of a Voltage Plane View

All voltage planes are presented the same way, with the same elements, as displayed below:

Figure 1-18 The voltage plane management bar

Table 1-4 Description of voltage plane management bar items

	Item name	Description
1	Channel list	Gives you access to the list of available channels. For more information, see “Setting Displayed Channel/C-scan Information” on page 78.
2	Channel label	Indicates the label given to a channel. For more information on labeling channels, see “Setting Displayed Channel/C-scan Information” on page 78.
3	Data type	Indicates the type of data displayed (channels, in the case of voltage planes).
4	Scale	Indicates the scale of the displayed data. For more information on modifying the scale, see “Measuring Indications” on page 261.
5	Frequency	Indicates the frequency of the displayed data. For more information on frequencies, see “Setting Displayed Channel/C-scan Information” on page 78.
6	Coil	Indicates the coil to which belongs the displayed data. For more information on coil numbers, see “Setting Displayed Channel/C-scan Information” on page 78.
7	Link	Allows you to link views. For more information, see “Linking Views” on page 61.

Figure 1-19 The voltage plane measurement bar

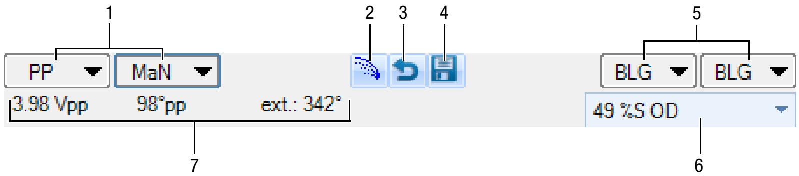
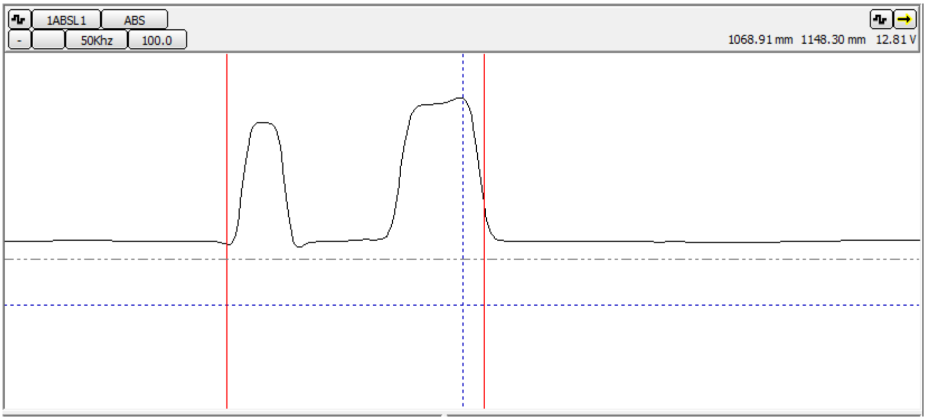


Table 1-5 Description of voltage plane measurement bar items

	Item name	Description
1	Measurement method selection	Allows you to choose a different measurement method for the signal displayed in the voltage plane. **PP:** peak-to-peak **PP FT:** peak-to-peak first transition **MR:** maximum rate **MaN:** manual For more information on measurement methods, see “Setting Amplitude/Phase Measurement Methods” on page 74.
2	Normalize	Normalizes the displayed signal (see “Editing the Voltage Plane Processing Unit” on page 156; see also “Performing RFT-Specific Tasks” on page 229).
3	Back to nominal	Returns the signal to the nominal point stored previously (see “Editing the Voltage Plane Processing Unit” on page 156; see also “Performing RFT-Specific Tasks” on page 229).
4	Save nominal	Saves the current adjustment to the nominal value (see “Editing the Voltage Plane Processing Unit” on page 156; see also “Performing RFT-Specific Tasks” on page 229).
5	Indication code assignment	Indicates the code that will be assigned to an indication when you click it.
6	Sizing curve selection	Displays the sizing curve view available for reporting.
7	Measurements	Measurements are displayed based on the measurement method selected. A new measurement has been added in Magnifi 3.1: **ext.**, which displays the circumferential extent of an indication.

Side View

Figure 1-20 Side view



Elements of a Side View

All side views are presented the same way, with the same elements, as displayed below:

Figure 1-21 The side view management bar

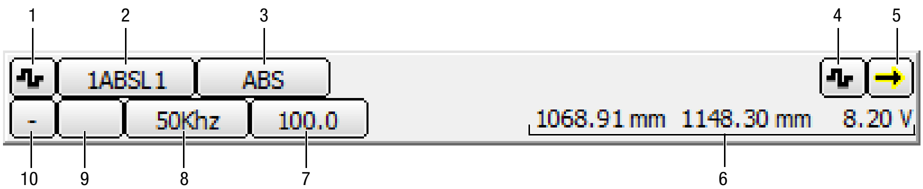
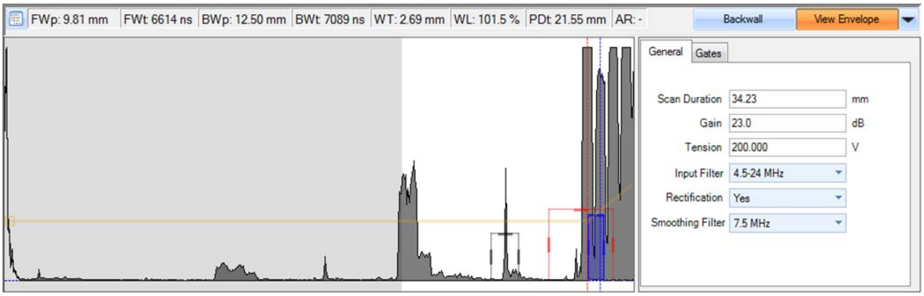


Table 1-6 Description of side view management bar items

	Item name	Description
1	Channel/ C-scan list	Gives you access to the list of available channels or C-scans. For more information, see "Setting Displayed Channel/C-scan Information" on page 78.
2	Channel/ C-scan name	Indicates the name of the channel or C-scan. For more information on naming channels, see "Setting Displayed Channel/C-scan Information" on page 78.
3	Channel/ C-scan label	Indicates the label given to a channel. For more information on labeling channels, see "Setting Displayed Channel/C-scan Information" on page 78.
4	Data type	Indicates the type of data displayed (C-scan X or Y component, or channels).
5	Axis	Indicates whether the vertical or horizontal component of the data is displayed.
6	Measurements	Gives the position of the vertical cursors and the amplitude of the horizontal cursors.
7	Scale	Indicates the scale of the displayed data. For more information on modifying the scale, see "Measuring Indications" on page 261.
8	Frequency	Indicates the frequency of the displayed data. For more information on frequencies, see "Setting Displayed Channel/C-scan Information" on page 78.
9	Coil	Indicates the coil to which belongs the displayed data. For more information on coil numbers, see "Setting Displayed Channel/C-scan Information" on page 78.
10	Link	Allows you to link views. For more information, see "Linking Views" on page 61.

A-scans

Figure 1-22 A-scan view



Elements of an A-scan

All A-scans are presented the same way, with the same elements, as displayed below:

Figure 1-23 The A-scan view management

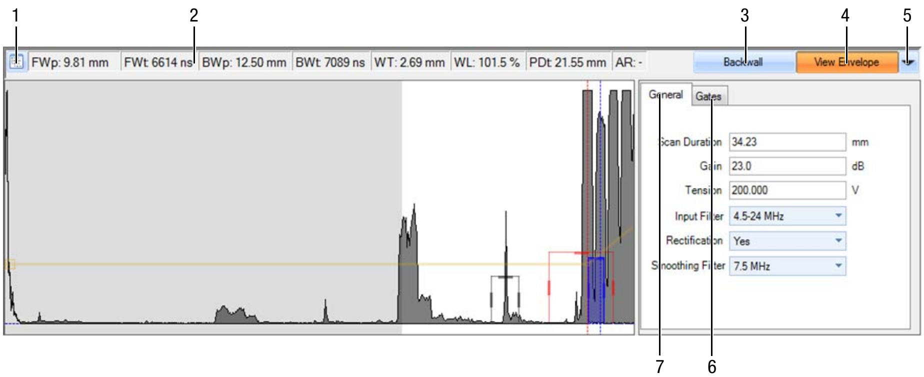


Table 1-7 Description of A-scan view items

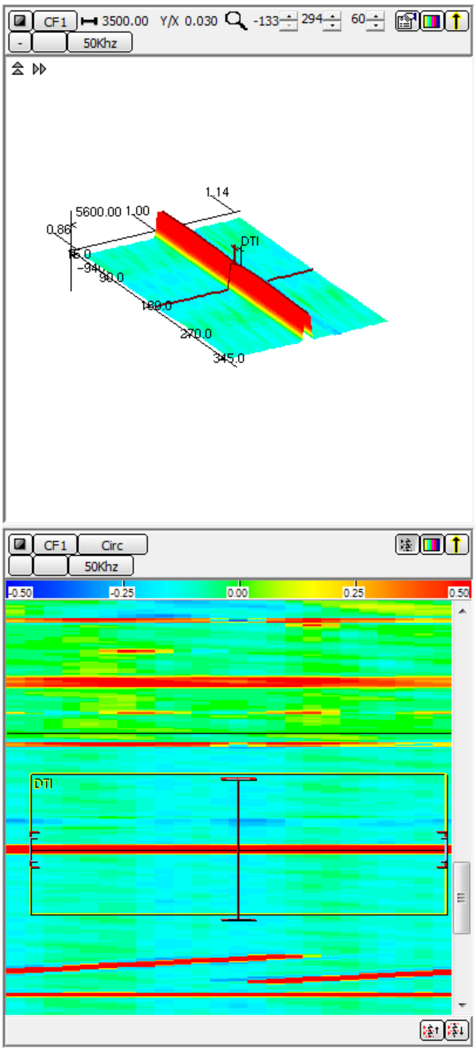
	Item name	Description
1	Information field customization button	Allows you to select the information fields that you want to display above the A-scan (for more information, see "Configuring Information Fields" on page 59).
2	Information fields	Information fields giving specific data that you want to monitor in real time (for more information, see "Configuring Information Fields" on page 59).
3	Backwall button	Zooms the A-scan time scale to show only the front wall and backwall echoes.
4	View Envelope button	Keeps the maximum values of each A-scan.
5	Display/Hide A-scan	When the A-scan is displayed, this button points down, indicating that clicking on it will send it at the bottom of the screen (hide), whereas, when it points up, it indicates that the A-scan is hidden, and that clicking it will bring up the A-scan.
6	General tab	Allows you to configure general A-scan parameters (see "Configuring A-scan View Parameters" on page 57).
7	Gates tab	Allows you to configure the detection gates (see "Configuring A-scan View Parameters" on page 57).

C-scans

Magnifi offers two types of C-scans:

- 3D
- Standard

Figure 1-24 3D C-scan (top) and standard C-scan (bottom)



Elements of a C-scan

All C-scan views (3D and standard) are presented the same way, with the same elements, as displayed below:

Figure 1-25 The 3D and standard C-scan management bar

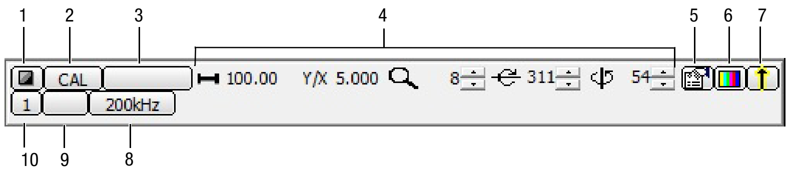


Table 1-8 Description of 3D and standard C-scan management bar items

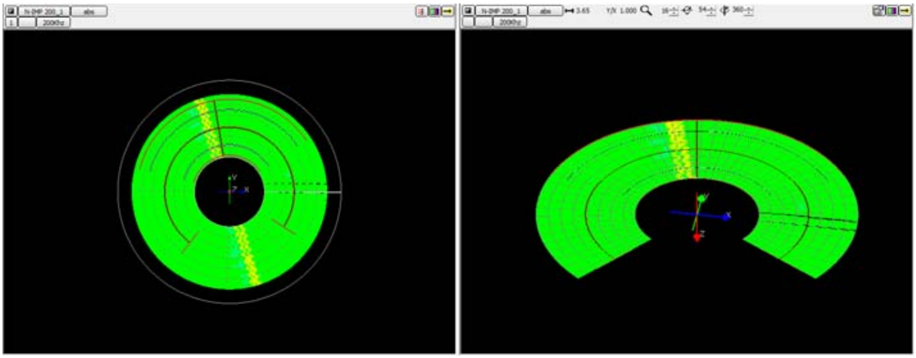
	Item name	Description
1	Channel/ C-scan list	Gives you access to the list of available channels or C-scans. For more information, see "Setting Displayed Channel/C-scan Information" on page 78.
2	Channel/ C-scan name	Indicates the name of the channel or C-scan. For more information on naming channels, see "Setting Displayed Channel/C-scan Information" on page 78.
3	Channel/ Labels	Allows you to select the type of channel that you are manipulating, raw, filtered, or unlabeled.
4	3D C-scan values (*3D C-scan **only**.)*	Allows you to view the values applied to the rotation of a 3D C-scan, and to modify a few of them (values with up-down arrows on their right).
5	3D C-scan tools button *(3D C-scan **only**.)*	Adds a line of management tools to change the parameters of the displayed 3D C-scan. In standard C-scans, this button is replaced with a **Display navigation panel** button that gives you access to controls for moving between indications.
6	Color palette	Allows you to display/hide the C-scan color palette. For more information, see "Customizing Color Schemes" on page 41.
7	Axis	Indicates whether the vertical or horizontal component of the data is displayed.
8	Frequency	Indicates the frequency of the displayed data. For more information on frequencies, see "Setting Displayed Channel/C-scan Information" on page 78.
9	Coil	Indicates the coil to which belongs the displayed data. For more information on coil numbers, see "Setting Displayed Channel/C-scan Information" on page 78.
10	Link	Allows you to link views. For more information, see "Linking Views" on page 61.

Polar Scans

Magnifi offers two types of polar scans:

- Standard
- 3D

Figure 1-26 Standard polar scan (left) and 3D polar scan (right)



Elements of a Polar Scan

All polar scans (3D and standard) are presented the same way, with the same elements, as displayed below:

Figure 1-27 The 3D and standard polar scan management bar

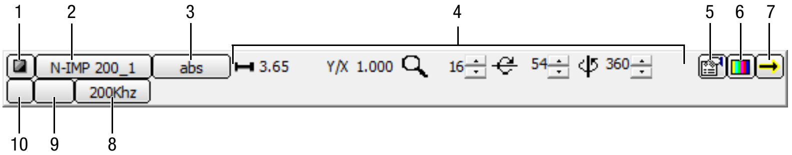
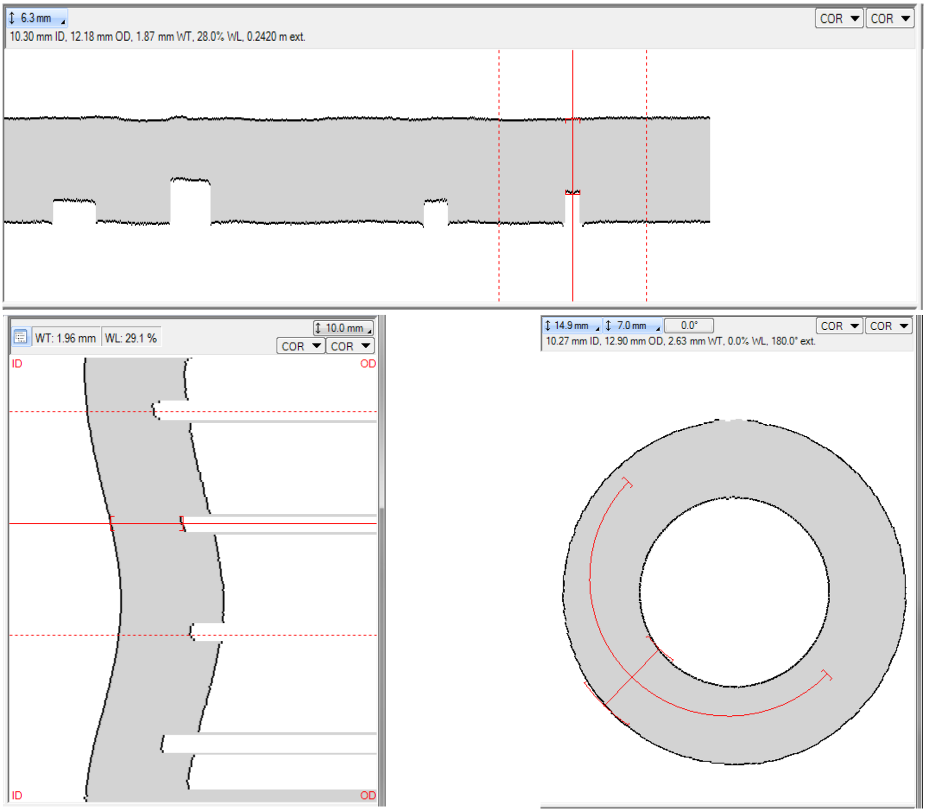


Table 1-9 Description of 3D and standard polar scan management bar items

	Item name	Description
1	Channel/ C-scan list	Gives you access to the list of available channels or C-scans. For more information, see "Setting Displayed Channel/C-scan Information" on page 78.
2	Channel/ C-scan name	Indicates the name of the channel or C-scan. For more information on naming channels, see "Setting Displayed Channel/C-scan Information" on page 78.
3	Channel/ C-scan label	Indicates the label given to a channel. For more information on labeling channels, see "Setting Displayed Channel/C-scan Information" on page 78.
4	3D polar scan values (*3D polar scans **only**.*)	Allows you to view the values applied to the rotation of a 3D polar scan, and to modify a few of them (values with up-down arrows to their right).
5	3D polar scan tools button *(3D polar scan **only**.)*	Adds a line of management tools to change the parameters of the displayed 3D polar scan. In standard polar scans, this button is replaced with a **Display navigation panel** button that gives you access to controls for moving between indications.
6	Color Palette	Allows you to display/hide the polar scan color palette. For more information, see "Customizing Color Schemes" on page 41.
7	Axis	Indicates whether the vertical or horizontal component of the data is displayed.
8	Frequency	Indicates the frequency of the displayed data. For more information on frequencies, see "Setting Displayed Channel/C-scan Information" on page 78.
9	Coil	Indicates the coil to which belongs the displayed data. For more information on coil numbers, see "Setting Displayed Channel/C-scan Information" on page 78.
10	Link	Allows you to link views. For more information, see "Linking Views" on page 61.

Projection Views

Figure 1-28 Projection views (top: longitudinal, left: circumferential, right: cylindrical)



Elements of a Projection View

Projection views are used specifically with IRIS inspections. All projection views are presented the same way, with the same elements, as displayed below:

Figure 1-29 The projection view management bar

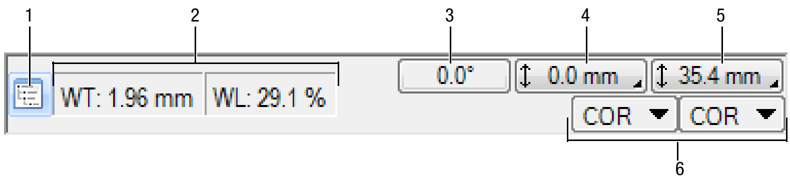


Table 1-10 Description of the projection view management bar items

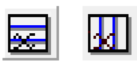
	Item name	Description
1	Information field customization button	Allows you to display or hide information fields relevant to your inspection project. For more information on information fields, see “Configuring Information Fields” on page 59.
2	Information fields	Information fields giving specific data that you want to monitor in real time (for more information, see “Configuring Information Fields” on page 59).
3	Rotation (*cylindrical view option **only***.)	Rotates the cylinder projection view.
4	Internal diameter reduction (*cylindrical view **only***.)	Changes the aspect ratio of the tube wall thickness compared to the outside diameter by reducing the internal diameter.
5	Scale	Indicates the scale of the displayed data. For more information on modifying the scale, see “Measuring Indications” on page 261.
6	Indication code assignment buttons	Indicates the code that will be assigned to an indication when you click it.

Dependent vs. Independent Panes

Panes in a window have certain specific behaviors that you should be aware of before you start creating layouts.

Dependent panes are created by using a Multi split (horizontally or vertically) button.

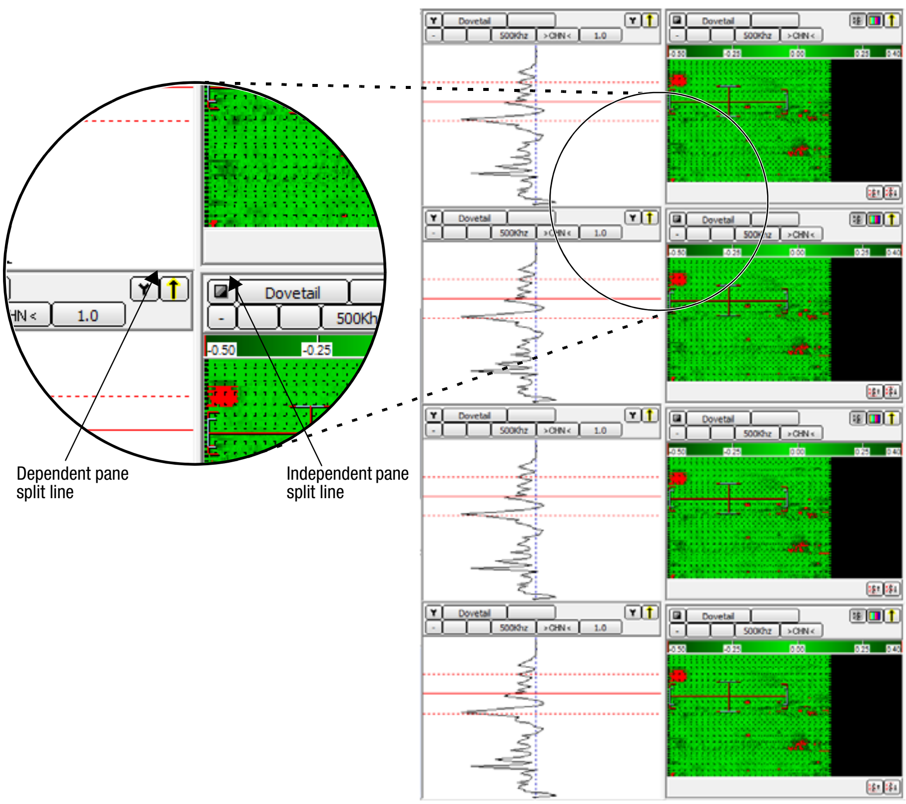
Figure 1-30 Multi split horizontally and Multi split vertically buttons



Dependent panes are characterized by two specific elements:

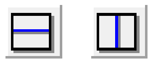
- The split line is thinner than a independent pane split line
- When you place the cursor over the split line, the cursor does not change to indicate that you can resize the pane.

Figure 1-31 Characteristics of dependent panes



Independent panes are created by using a Split (horizontally or vertically) button.

Figure 1-32 Split horizontally and Split vertically buttons



Independent panes are characterized by two specific elements:

- The split line is thicker than a dependent pane split line
- When you place the cursor over the split line, the cursor changes to indicate that you can resize the pane.

Creating Layouts

There are four ways to create layouts in Magnifi:

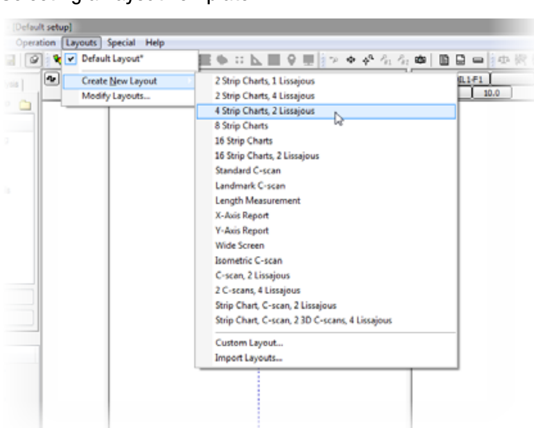
- From a template,
- From scratch,
- By duplicating an existing layout before modifying it,
- By importing layouts from an existing setup file.

From a Template

To create a new layout from a template:

1. On the **Layouts** menu, point to **Create New Layout**, and then click a template in the list. These templates are named by the elements they contain (e.g., **4 Strip Charts, 2 Lissajous**).

Figure 1-33 Selecting a Layout Template



2. Select the template on which you want to base your new layout. The layout on-screen changes based on your selection.

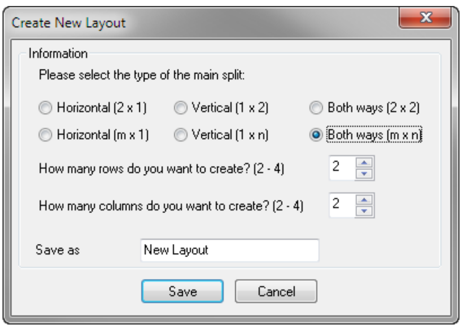
From there, you can change the layout as much as you need to, and rename it as well, if needed. The procedures to modify a layout and to rename it are explained in the following pages.

From Scratch

To create a completely new layout:

1. From the **Layouts** menu, select **Create New Layout>Custom Layout**. The **Create New Layout** dialog box appears.

Figure 1-34 The **Create New Layout** dialog box



2. Select a main split:
   - Predefined: **Horizontal (2×1), Vertical (1×2),** or **Both ways (2×2)**
   - Custom: **Horizontal (m×1), Vertical (1×m),** or **Both ways (m×n).** The “**How many rows/columns...**” lines appear only when you select a custom main split.
3. If you selected a custom main split, enter the number of rows and/or columns that you want in the proper dialog boxes.
4. Enter a new name for the layout in the **Save as** text box, and click **Save**. The name of the new layout now appears and is selected in the **Layouts** menu.

When you create a new layout, each pane is empty and shows a circle with **Control missing** in its center. To assign a view to a pane, see “Assigning a View to a Pane” on page 43.

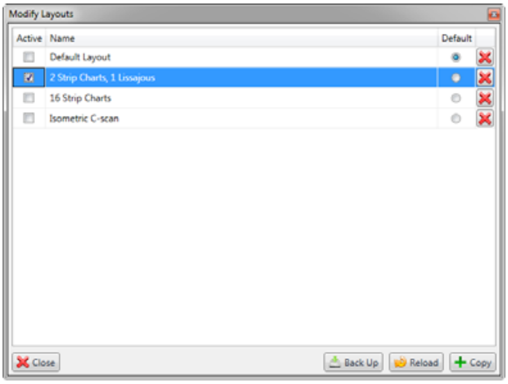
Duplicating Layouts

You can duplicate an existing layout, if needed.

To do so:

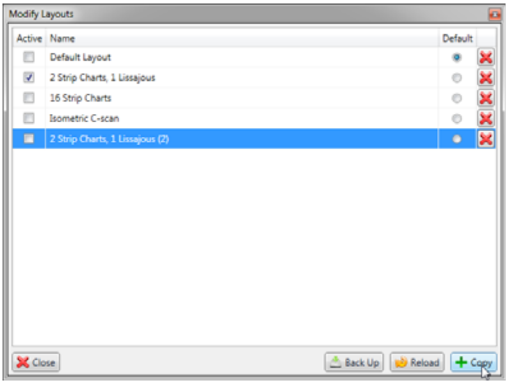
1. From the **Layouts** menu, select **Modify Layouts.** The **Modify Layouts** window appears.

Figure 1-35 The **Modify Layouts** window



2. From the list of layouts, highlight the layout the you want to duplicate.
3. Click **Copy.** The highlighted layout is copied and added to the list with the name of the original layout followed by a number in parentheses.

Figure 1-36 Copied layout



You can rename the layout as you see fit after it has been copied.

Importing Layouts from an Existing Setup

If you already have a setup file containing the layouts that you want to use, you can import these layouts in a new setup.

To do so:

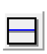
1. From the **Layouts** menu, select **Create New Layout>Import Layouts.**
2. In the **Import Layouts** window that opens, click the Browse button.
   A standard **Open** dialog box appears.
3. Browse to find the setup file containing the layouts that you want to import and click **Open.** The layouts available in the selected setup file appear in the **Layouts** list of the **Import Layouts** window.
4. Click **Import.** The listed layouts are now imported in your current setup file.

Customizing Layouts

Magnifi allows you to customize layouts to perfectly meet your needs. The customizing procedures are explained in the following pages.

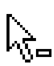
Splitting Panes Horizontally in Two

To split a pane horizontally in two:



1. From the Layout toolbar, click the **Split horizontally** button.
2. Position the cursor over the pane that you want to split in two. The cursor changes to this:

Figure 1-37 The Split Horizontal cursor



3. Click in the pane. The pane is split horizontally in two, with the same data displayed in both panes.

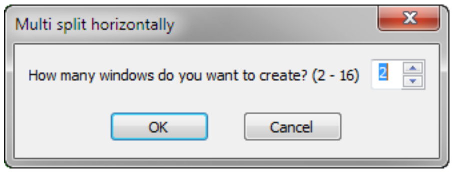
Note *Panes created with a Split button (horizontally or vertically) are independent, resizable, and the view in each can be modified.*

Splitting Panes Horizontally in More Than Two

To split a pane horizontally in more than two:

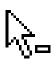
1. From the Layout toolbar, click the **Multi split horizontally** button. The **Multi split horizontally** window appears.

Figure 1-38 The **Multi split horizontally** window



2. Enter the number of panes that you want to create, between 2 and 16, and click **OK.**
3. Position the cursor over the pane that you want to split. The cursor changes to this:

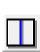
Figure 1-39 The Split Horizontal cursor



4. Click in the pane. The pane is split horizontally in the number of panes that you entered in the **Multi split horizontally** window.

Note *Panes created with the Multi split button (horizontally or vertically) are NOT resizable, are dependent on the original pane, and all show the same view.*

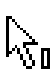
Splitting Panes Vertically in Two



To split a pane vertically in two:

1. From the Layout toolbar, click the **Split vertically** button.
2. Position the cursor over the pane that you want to split in two. The cursor changes to this:

Figure 1-40 The Split Vertical cursor



3. Click in the pane. The pane is split vertically in two, with the same data displayed in both panes.

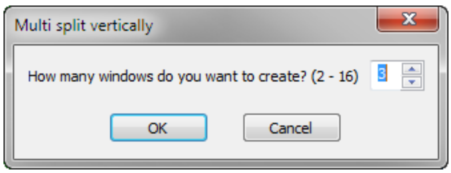
Note *Panes created with a Split button (horizontally or vertically) are independent, resizable, and the view in each can be modified.*

Splitting Panes Vertically in More Than Two

To split a pane vertically in more than two:

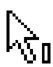
1. From the Layout toolbar, click the **Multi split vertically** button. The **Multi split vertically** window appears.

Figure 1-41 The **Multi split vertically** window



2. Enter the number of panes that you want to create, between 2 and 16, and click **OK.**
3. Position the cursor over the pane that you want to split. The cursor changes to this:

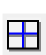
Figure 1-42 The Split Vertical cursor



4. Click in the pane. The pane is split vertically in the number of panes that you entered in the **Multi split vertically** window.

Note *Panes created with the Multi split button (horizontally or vertically) are NOT resizable, are dependent on the original pane, and all show the same view.*

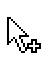
Splitting Panes in Four



To split a pane in four:

1. From the Layout toolbar, click the **Split in both directions** button.
2. Position the cursor over the pane that you want to split. The cursor changes to this:

Figure 1-43 The Split in both directions cursor



3. Click in the pane. The pane is split in four, with the same data displayed in all panes.

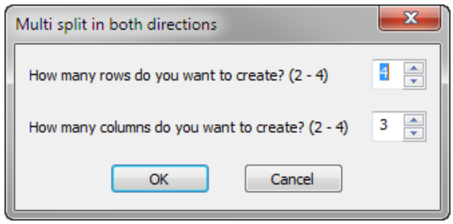
Note *Panes created with a Split button (horizontally, vertically, or in both directions) are independent, resizable, and the view in each can be modified.*

Splitting Panes in Any Number of Rows and Columns

To split a pane in any number of rows and columns:

1. From the Layout toolbar, click the **Multi split in both directions** button. The **Multi split in both directions** window appears.

Figure 1-44 The **Multi split in both directions** window



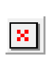
2. In the first text box, enter the number of rows that you want to create, between 2 and 4.
3. In the second text box, enter the number of columns that you want to create, between 2 and 4.
4. Position the cursor over the pane that you want to split. The cursor changes to this:

Figure 1-45 The Split in both directions cursor

5. Click in the pane. The pane is split vertically and horizontally in the number of panes that you entered in the **Multi split in both directions** window.

Note *Panes created with the Multi split button (horizontally, vertically, or in both directions) are NOT resizable, are dependent on the original pane, and all show the same view.*

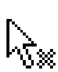
Removing Panes



To remove panes from a window:

1. From the Layout toolbar, select the **Remove the custom layout** button.
2. Position the cursor over the pane that you want to remove. The cursor changes to this:

Figure 1-46 The Remove Pane cursor



3. Click in the pane to remove.

If you clicked on an *independent* pane, only this pane is removed. If you removed a horizontal pane, the independent pane underneath fills the empty space. If you removed a vertical pane, the independent pane to the right fills the empty space.

If you clicked on a *dependent* pane, *all* panes associated to the original pane (the independent pane that existed before the multi split) are removed, save for the original pane. If you removed a horizontal pane, the independent pane underneath fills the space left empty. If you removed a vertical pane, the independent pane to the right fills the space left empty.

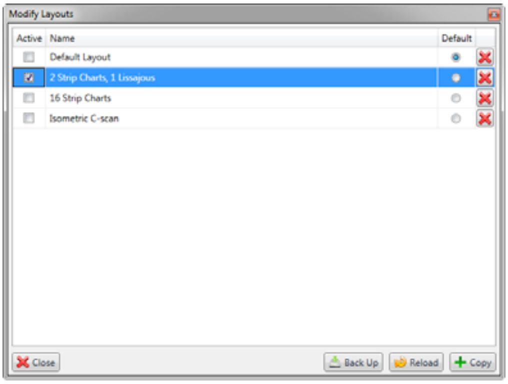
Specifying a Default Layout

When Magnifi starts, it loads its default layout. You can specify a different default layout as the one under the default Magnifi configuration.

To do so:

1. Design a layout as needed; you can also select one from the **Create New Layout** sub-menu.
2. Select **Modify Layout** from the **Layouts** menu. The **Modify Layouts** window appears.
3. In the window, all recently used layouts are listed. In the **Default** column, select the layout that you want to use as default.

Figure 1-47 The **Modify Layouts** window



From now on, the layout that you have designed will load automatically as default when you launch Magnifi.

Removing Layouts from the List

To remove layouts from the list:

1. Select **Modify Layout** from the **Layouts** menu. The **Modify Layouts** window appears (see Figure 1-47).
2. Click the red X next to the layout name that you want to remove from the list. The layout is removed from the list.

Managing Views

Magnifi allows you to decide how your acquired data will be displayed.

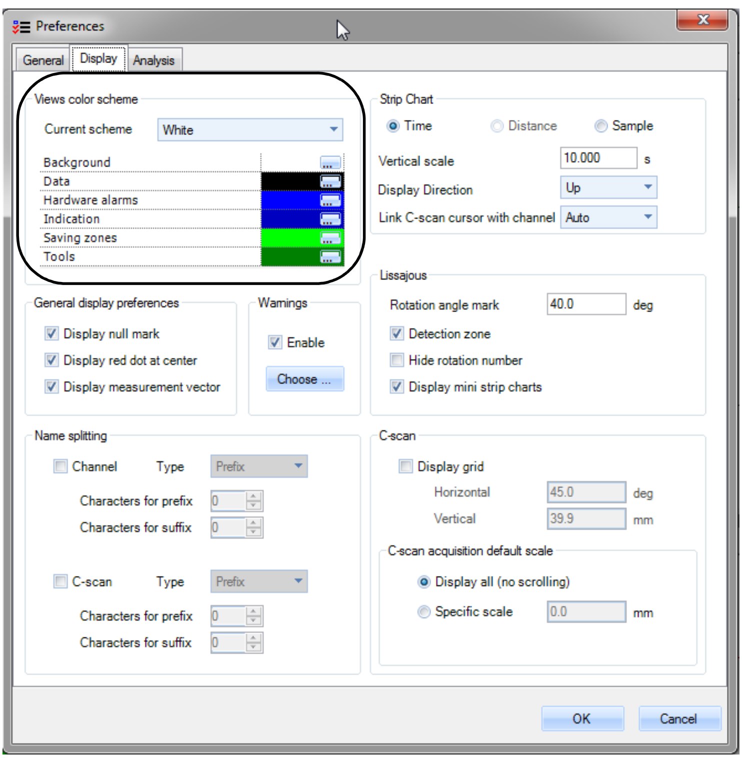
Configuring Views Colors

Magnifi comes standard with five preset color schemes. Color schemes affect the colors used in the data display area.

To select a color scheme:

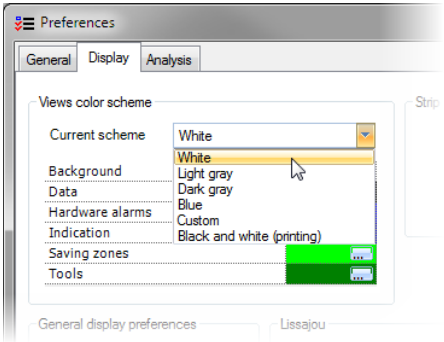
1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. Click the **Display** tab.

Figure 1-48 The **Display** tab



3. In the **Views color scheme** section, select from the **Current scheme** drop-down list the color scheme that you want to use. **White** is selected by default. When you click a color scheme, the colors underneath the drop-down list change.

Figure 1-49 Choosing a color scheme



4. Click **OK.** The selected color scheme is applied to the views in the current layout.

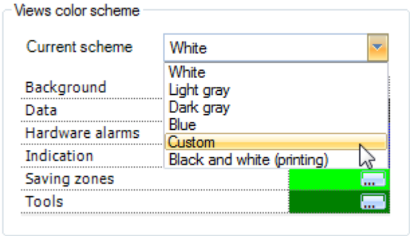
Customizing Color Schemes

If none of the existing color schemes found in the **Current scheme** drop-down list meet your requirements, you can create your own.

To create a custom color scheme:

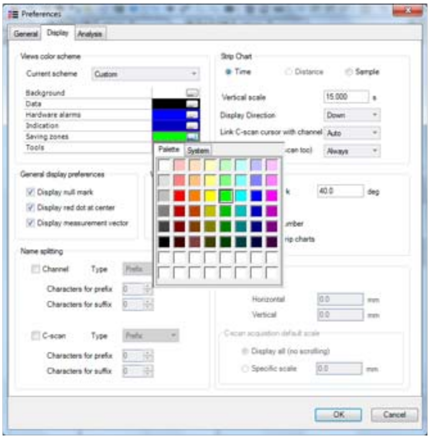
1. From the **Current scheme** drop-down list, select **Custom.**

Figure 1-50 Selecting the **Custom** color scheme



2. Click the “...” button next to the item that you want to customize. Two tabs appear where you can select colors: the **Palette** and the **System** tabs (see Figure 1-51).

Figure 1-51 Custom color tabs



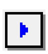
3. Click the color that you want. When you release the mouse button, the color that you clicked replaces the previous color.
4. Repeat step 3 for all the items that you want to customize.
5. When you are finished customizing your color scheme, click **OK.** The **Preferences** dialog box closes and your custom color scheme is applied.

Assigning a View to a Pane

Magnifi offers you the following views to assign to any pane:

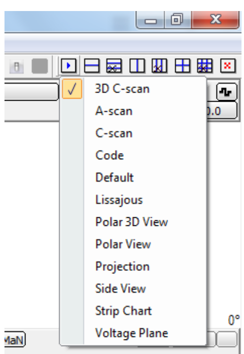
- **3D C-Scan**
- **A-Scan**
- **C-Scan**
- **Code**
- **Default**
- **Lissajous**
- **Polar 3D View**
- **Polar View**
- **Projection**
- **Side View**
- **Strip Chart**
- **Voltage Plane**

To assign a view to a pane:



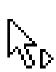
1. In the Layout toolbar, click the **Replace view** button. A list of available views appears.

Figure 1-52 List of available views



2. Select the view that you want to assign. The menu closes and the cursor changes to this:

Figure 1-53 The Assign View cursor



3. Click in the pane where you want to assign the selected view. The view appears in the pane.

Configuring General Strip Chart Parameters

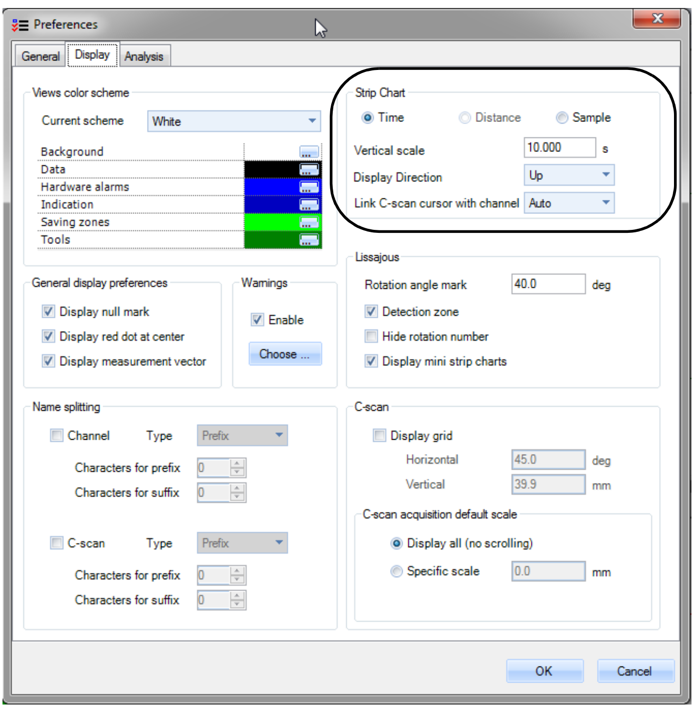
Strip charts display inspection results, generally over a time period and from bottom to top of the screen.

In Magnifi, you can configure parameters that apply to all displayed strip charts (measurement unit and display direction of strip charts). You can also configure parameters that apply to each strip chart individually (see page 46).

To do so:

1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. Click the **Display** tab.

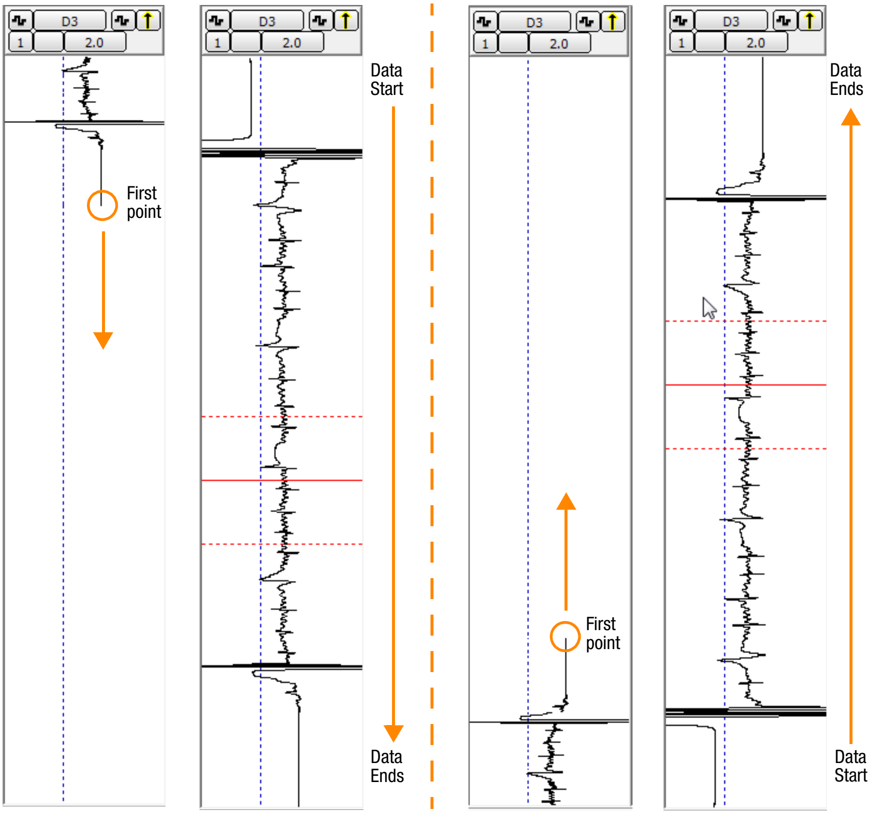
Figure 1-54 The **Display** tab



3. In the **Strip Chart** section, select whether you want strip charts to display a time scale, a distance scale, or a sample scale (affects display only during acquisitions).
4. In the **Vertical scale** text box, enter the scale steps in seconds, millimeters (or inches), or number of samples.

5. In the **Display Direction** drop-down list, select **Up** or **Down.**
   - If you select **Up:** during acquisition, the first data point is displayed at the bottom of the screen and the following points are placed above the first point. After the acquisition, the beginning of the acquisition (first data seen by the probe) is at the bottom of the screen and the end of the tube is at the top. Visually, the data is read from bottom to top.
   - If you select **Down:** during acquisition, the first data point is displayed at the top of the screen and the following points are placed under the first point. After the acquisition, the beginning of the acquisition (first data seen by the probe) is at the top of the screen and the end of the tube is at the bottom. Visually, data is read from top to bottom.

Figure 1-55 Examples of data display directions (down: left; up: right)



6. In the **Link C-scan cursor with channel** drop-down list, select **Auto, Yes,** or **No:**
   - If you select **Auto,** channel and C-scan cursor positions are automatically linked when and if possible. Cases where it would not be possible is if you are using a raster or polar scan, or performing a back-and-forth inspection with an encoder.
   - If you select **Yes,** it forces the position of the channel and C-scan cursors to be the same. This is useful when you display both types of views in the same layout. This option does not work for raster scan types.
   - If you select **No,** channel and C-scan cursor positions are kept independent.

Configuring Individual Strip Chart Parameters

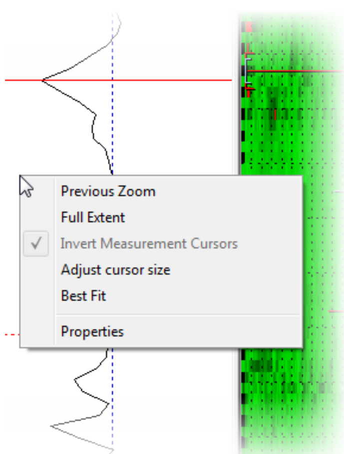
Strip charts display inspection results, generally over a time period and from bottom to top of the screen.

You can configure parameters that apply to all displayed strip charts (measurement unit and display direction of strip charts) (see page 44). In Magnifi, you can also configure parameters that apply to each strip chart individually.

To do so:

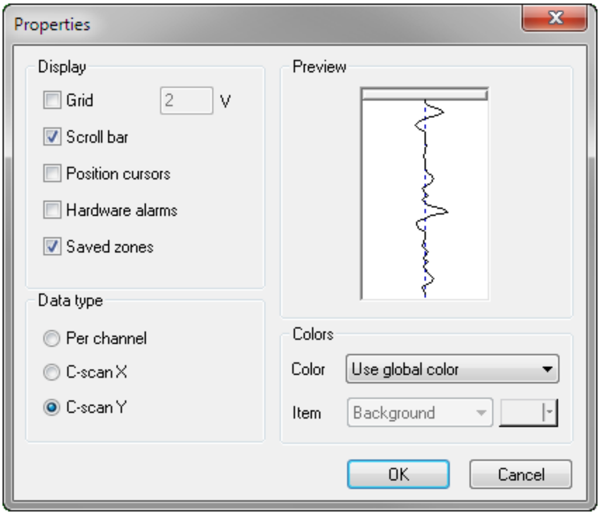
1. Right-click over a displayed strip chart. A contextual menu appears.

Figure 1-56 Strip chart contextual menu



2. Select **Properties.** The **Properties** window for this strip chart appears. In the **Preview** section, you can see what will happen to the displayed data as you select to display elements and change colors.

Figure 1-57 **Properties** window for a specific strip chart



3. In the **Display** section, you can choose to display or hide:
   - **Grid:** a grid (with accompanying voltage scale).
   - **Scroll bar:** a scroll bar specific to the selected strip chart, visible when zoomed in on a part of the chart.
   - **Position cursors:** a vertical and a horizontal line that follow the cursor as you move it over the strip chart.
   - **Hardware alarms:** the level of hardware alarms (not supported by the Ectane at this time).
   - **Saved zones:** a green background indicating the data recorded in acquisition mode.
4. In the **Data type** section, you can select the type of data displayed, either from channels (i.e., for bobbin probes) or from C-scan axes (i.e., from array probes).
5. In the **Colors** section, you can choose the colors that you want to use for the strip chart. These colors are the same as the ones defined in "Configuring Views Colors" on page 40.
6. Click **OK** when you are done. The selected options now appear in the strip chart.

Configuring General Lissajous View Parameters

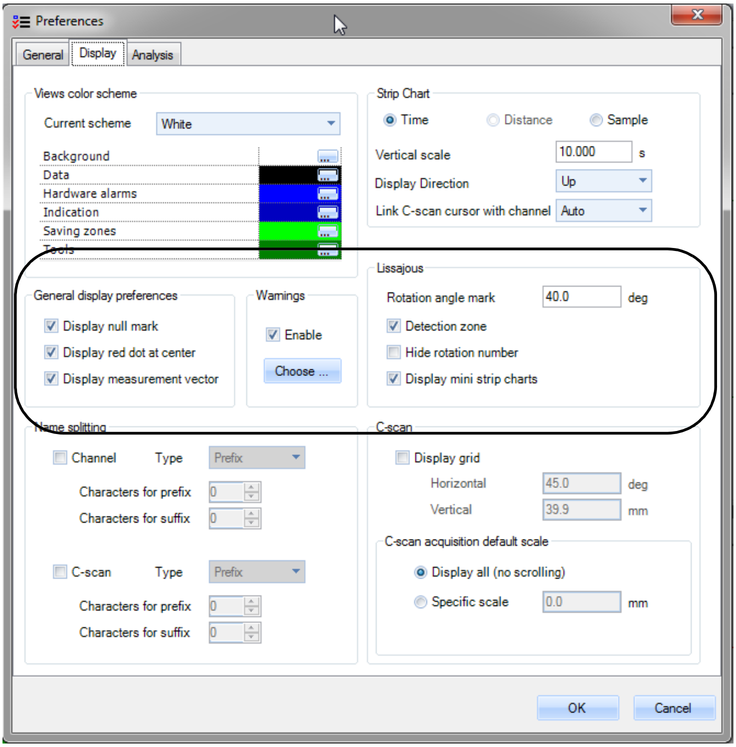
Lissajous provide information not easily available from other types of views. Some information can be hidden or displayed during the acquisition or when performing analysis. In Magnifi, you can configure parameters that apply to each Lissajous individually (see page 51) or to all displayed Lissajous.

The following procedure explains what are the tools and how to activate/deactivate them for all Lissajous.

To configure Lissajous:

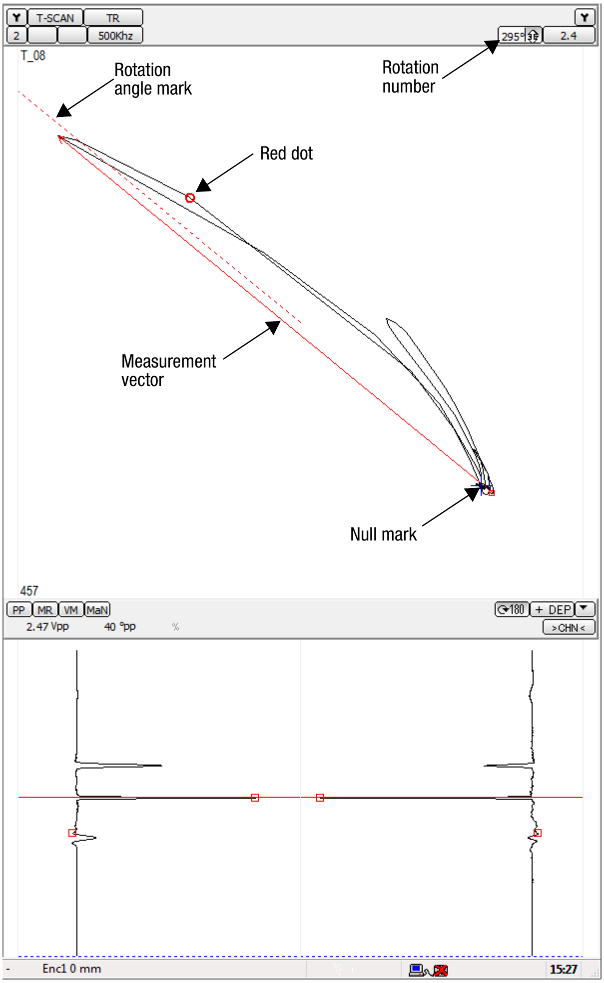
1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. Click the **Display** tab.

Figure 1-58 The **Display** tab



3. In the **General display preferences** section, select the measurement tools (see Figure 1-59) that you want to display by checking the appropriate boxes:
   a. To display the null mark (a blue cross), check the **Display null mark** box.
   b. To display a red dot corresponding to the data selection cursor position, check the **Display red dot at center** box.
   c. To display the measurement vector when automatic measurements are active, check the **Display measurement vector** box.
4. In the **Lissajous** section, you can specify certain rotation parameters:
   a. in the **Rotation angle mark** text box, enter the required angle at which you want to rotate your signal to better see all the relevant information. The default value is 40°. The mark appears when you rotate the Lissajous (see Figure 1-59).
   b. You can also prevent the rotation number from appearing in the channel/ C-scan management bar by selecting **Hide rotation number**. This will hide the rotation number control, thus preventing rotation of the Lissajous directly on-screen. Manual rotation can still be performed by pressing CTRL+left-click-and-drag.
5. When you are done configuring the Lissajous display, click **OK.**

Figure 1-59 Lissajous with measurement tools



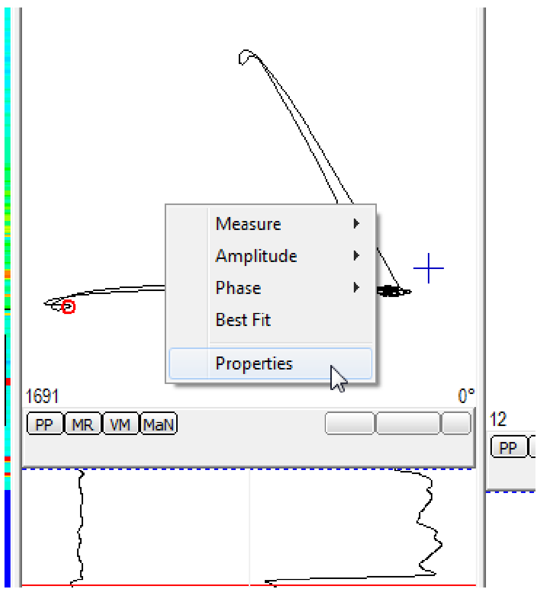
Configuring Individual Lissajous View Parameters

In Magnifi, you can configure parameters that apply to all displayed Lissajous (see page 48) or to each Lissajous individually.

To do so:

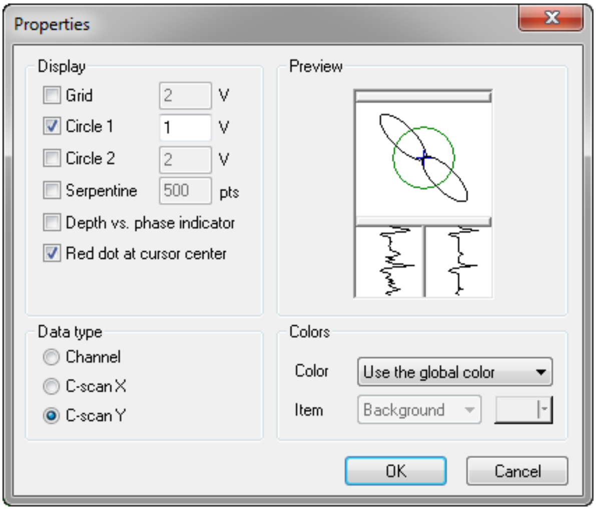
1. Right-click over a displayed Lissajous. A contextual menu appears.

Figure 1-60 Lissajous contextual menu



2. Select **Properties.** The **Properties** window for this Lissajous appears. In the **Preview** section, you can see what will happen to the displayed data as you select to display elements and change colors.

Figure 1-61 **Properties** window for selected Lissajous



3. In the **Display** section, you can choose to display or hide:
   - **Grid:** a square grid (with accompanying voltage scale).
   - **Circle 1, Circle 2:** a visual aid representing a voltage scale around the central point.
   - **Serpentine:** number of points displayed during an acquisition; older points are deleted.
   - **Depth vs. phase indicator:** in cases where the sizing curve is of the phase-type, allows you to display visual aids to help correlate angles and depths (**O:** outside, **I:** inside).
   - **Red dot at cursor center:** indicate with a red dot the position of the data selection cursor on the equivalent strip chart.
   - **Show strip chart:** displays mini strip charts at the bottom of the Lissajous.
4. In the **Data type** section, you can select the type of data displayed, either from channels (i.e., for bobbin probes) or from C-scan axes (i.e., from array probes).
5. In the **Colors** section, you can choose the colors that you want to use for the strip chart. These colors are the same as the ones defined in “Configuring Views Colors” on page 40.
6. Click **OK** when you are done. The selected options now appear in the Lissajous.

Configuring Voltage Plane Views

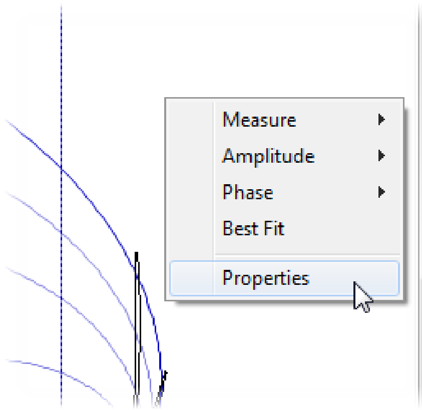
Voltage plane displays inspection results similar to a Lissajous in that it represents on two axes the vertical and horizontal components of a signal, but it also adds a skin depth attenuation curve that is very helpful when analyzing RFT inspection data.

In Magnifi, you can configure parameters that apply to each voltage plane individually.

To do so:

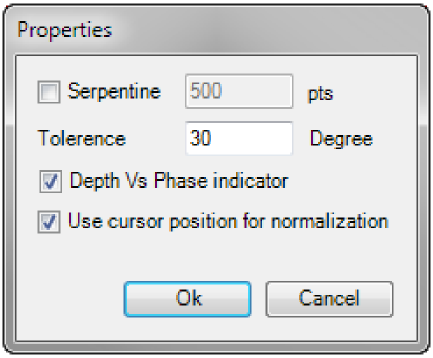
1. Right-click over a displayed voltage plane. A contextual menu appears.

Figure 1-62 Voltage plane contextual menu



2. Select **Properties.** The **Properties** window for this voltage plane appears.

Figure 1-63 Voltage plane **Properties** window



3. **Serpentine:** indicate the number of points displayed during an acquisition; older points are deleted.
4. **Tolerance:** indicates a threshold from which angle corrections (following normalization) are displayed in red.
5. **Depth vs. phase indicator:** in cases where the sizing curve is of the phase-type, allows you to display visual aids to help correlate angles and depths (**O:** outside, **I:** inside).
6. **Use cursor position for normalization:** allows you to perform a normalization by using only the data selection brackets (applies only if data is acquired without balancing).
7. Click **OK** when you are done. The selected options now appear in the voltage plane.

Configuring Side Views

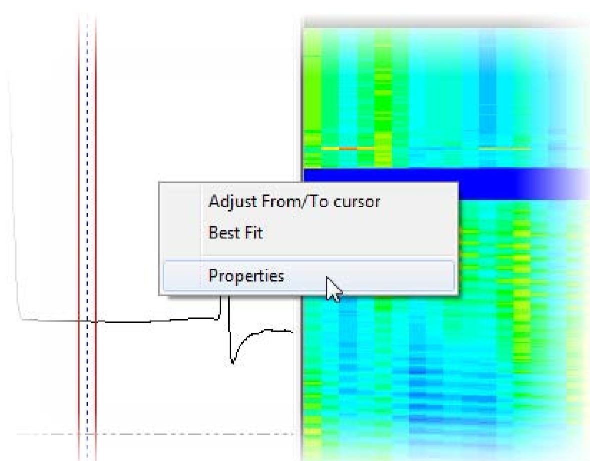
Side views displays inspection results horizontally rather than vertically (strip chart)

In Magnifi, you can configure parameters that apply to each side view individually.

To do so:

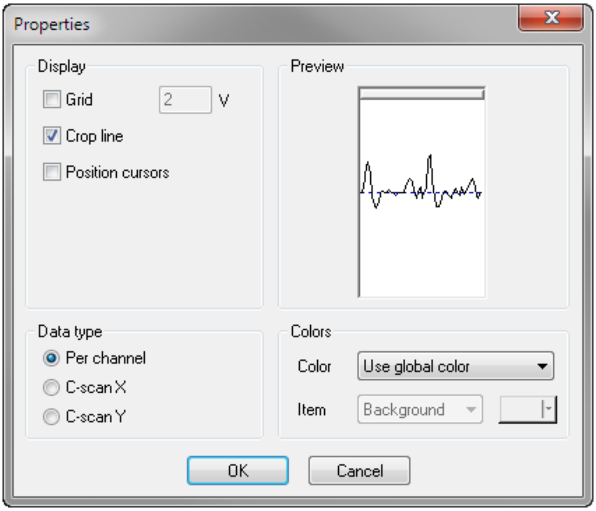
1. Right-click over a displayed side view. A contextual menu appears.

Figure 1-64 Side view contextual menu



2. Select **Properties.** The **Properties** window for this side view appears.

Figure 1-65 Side view **Properties** window



3. In the **Display** section, you can choose to display or hide:
   - **Grid:** a grid (with accompanying voltage scale).
   - **Crop line:** displays a horizontal cursor that serves as a threshold when measuring defects.
   - **Position cursors:** a vertical and a horizontal line that follow the cursor as you move it over the chart.
4. In the **Data type** section, you can select the type of data displayed, either from channels (i.e., for bobbin probes) or from C-scan axes (i.e., from array probes).
5. In the **Colors** section, you can choose the colors that you want to use for the strip chart. These colors are the same as the ones defined in “Configuring Views Colors” on page 40.
6. Click **OK** when you are done. The selected options now appear in the side view.

Configuring A-scan Views

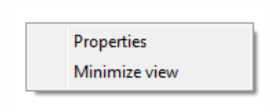
A-scans are a simple representation of an ultrasound scan performed during an IRIS inspection. They show the data detected as well as the measurement values for each data point.

In Magnifi, you can configure certain parameters that apply to each data point individually.

To do so:

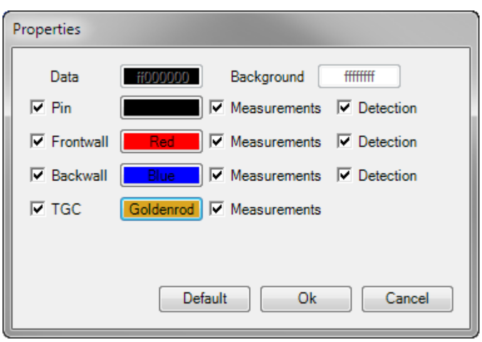
1. Right-click over a displayed A-scan view. A contextual menu appears.

Figure 1-66 A-scan contextual menu



2. Select **Properties.** The **Properties** window appears.

Figure 1-67 A-scan **Properties** window



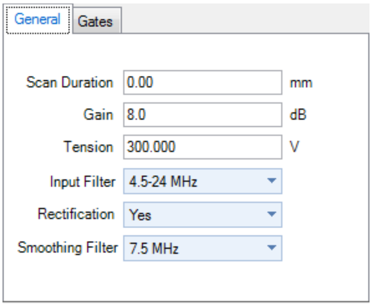
3. Check the box next to the visual tool that you want to display in the A-scan (pin gate, front wall gate, backwall gate, TGC slope).
4. For each displayed visual tool, select **Measurements** if you want to display measurement values for every time you modified the selected gate.
5. Also, for each displayed visual tool (except for the TGC), select **Detection** if you want to display a vertical line indicating the detected echo.
6. Click **OK** when you are done. The selected options now appear in the A-scan.

Configuring A-scan View Parameters

All A-scan views come with two tabs that allow modification of some general settings and all gate setting. These settings apply as soon as entered.

To setup general A-scan view parameters:

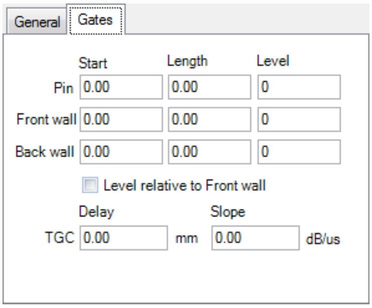
Figure 1-68 **General** A-scan view parameters



1. Set the length of the scan in the **Scan Duration** field (in mm or .in, depending on preferences.)
2. Enter the gain to apply in the **Gain** field.
3. Enter the excitation voltage in the **Tension** field.
4. Select an input filter from the **Input Filter** drop-down list.
5. Choose to apply signal rectification or not in the **Rectification** drop-down list.
6. Select your smoothing filter in the **Smoothing Filter** drop-down list.

To setup A-scan gate parameters:

Figure 1-69 **Gates** A-scan view parameters



1. Enter the **Start, Length**, and **Level** for the pin, front wall and back wall. If you want all levels to be relative to the front wall, check the **Level relative to Front wall** box.
2. Enter the delay and slope of the TGC in the appropriate fields.

Configuring Projection Views

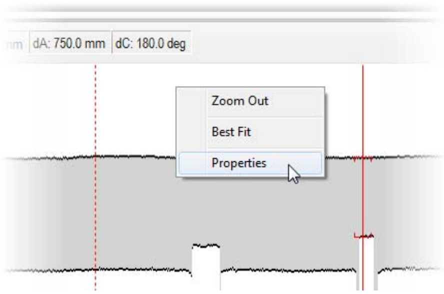
Projection views are used with the IRIS inspection technique only. They represent the wall thickness of the tube along the different axes.

In Magnifi, you can configure parameters that apply to each projection view individually.

To do so:

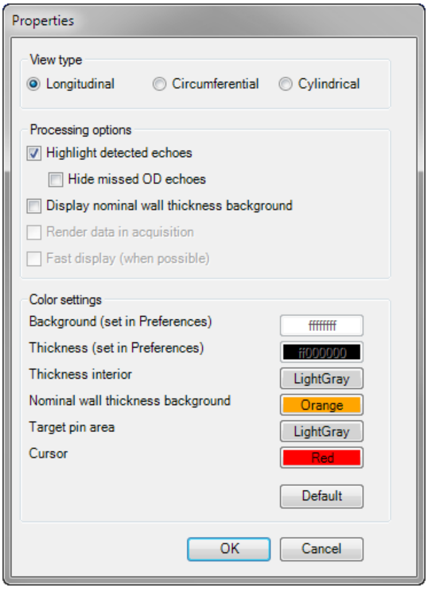
1. Right-click a displayed projection view. A contextual menu appears.

Figure 1-70 Projection view contextual menu



2. Click **Properties.** The **Properties** dialog box for this projection view appears.

Figure 1-71 Projection view **Properties** dialog box



3. In the **View type** section, select the projection view type that you want based on the tube axis that you want to represent.
4. Select the desired processing options:
   a **Highlight detected echoes:** gives a different color to the material inside the wall thickness compared to the detected internal and external echoes, that remain black. You can also choose to **Hide missed OD echoes** by selecting the check box (which hides the part of signals going beyond the outer limit of the tube where external OD echoes are not detected).
   b **Display nominal wall thickness background:** displays the theoretical position of the tube behind the data.
   c **Render data in acquisition:** displays data in real time.
5. Select the desired color settings for the various data elements displayed. The background and thickness are set in the **Preferences** window (see "Customizing Color Schemes" on page 41). For the other elements, click the color indicated. Color palettes appear. Click on the color that you want. The color palettes disappear and the new color is selected.

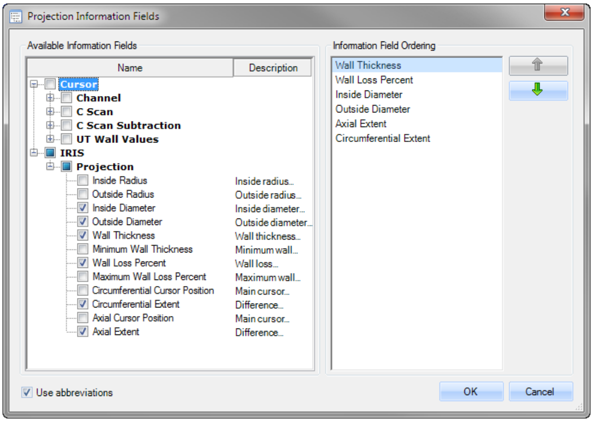
Configuring Information Fields

Information fields are available in both projection and A-scan views. Information fields give you immediate contextual values for each view in which they are displayed. They apply to IRIS inspections and cursor positions. For IRIS inspections, only projection values can be displayed. For cursor positions, you can display various channel, C-scan, C-scan subtraction and UT wall values.

To display information fields:

1. From a projection view or an A-scan view, click the Information Field customization button. The **AScanVideo Information Fields** or **Projection Information Fields** window appears. Their content is similar.

Figure 1-72 Information Fields window

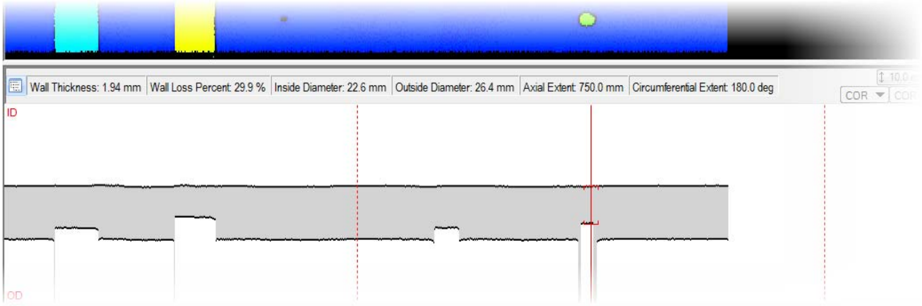


2. In the **Available Information Fields** section, check the boxes for the information fields that you want to display. They appear in the **Information Field Ordering** list.
3. From that list, you can order the fields using the arrow buttons. The first information field is the list will appear to the left-hand side of the view when you click **OK.**

Note *If you uncheck an information field in the **Available Information Fields** list, it is removed from the **Information Fields Ordering** column. If you check that information field again, it is added at the end of the **Information Fields Ordering** list.*

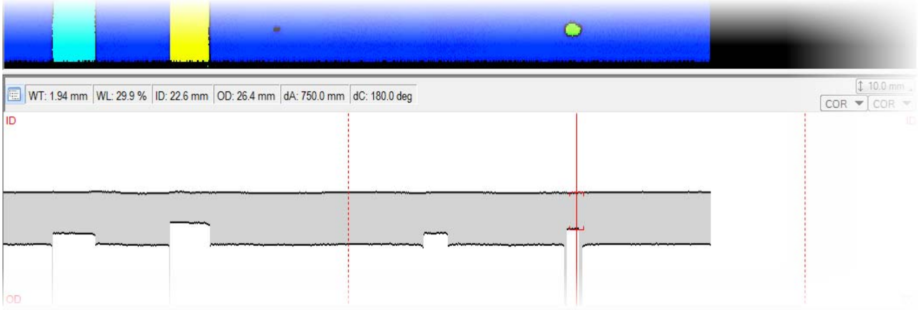
4. Click **OK.** The **Information Fields** window disappears and the selected information fields appears in the proper view.

Figure 1-73 Projection view with displayed information fields



Note *You can also display the information field titles with abbreviations. To do so, make sure to check the* ***Use abbreviations*** *box in the bottom left of the* ***Information Fields*** *window. This will allow you to display more information fields at a time.*

Figure 1-74 Projection view with abbreviated information fields



Linking Views

Once you have laid out your different views in Magnifi, you can link some or all of those views in such a way that modifying a view modifies all views linked to it.

To link views, click the link button (see “Understanding Basic Layout Elements” on page 11) in each of the views that you want to link so that they all show the same number. All views with the same “link” number are linked.

When views are linked, changes in the following are replicated in all linked views:

- Displayed channel/C-scan
- Channel name
- Channel label
- Data type

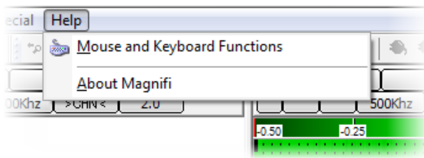
- Data component (horizontal, vertical, channel)
- Scale
- Frequency
- Coil

Displaying Mouse and Keyboard Functions

Magnifi offers many mouse and keyboard functions. If you need help remembering all these functions, you can easily review them directly from the program.

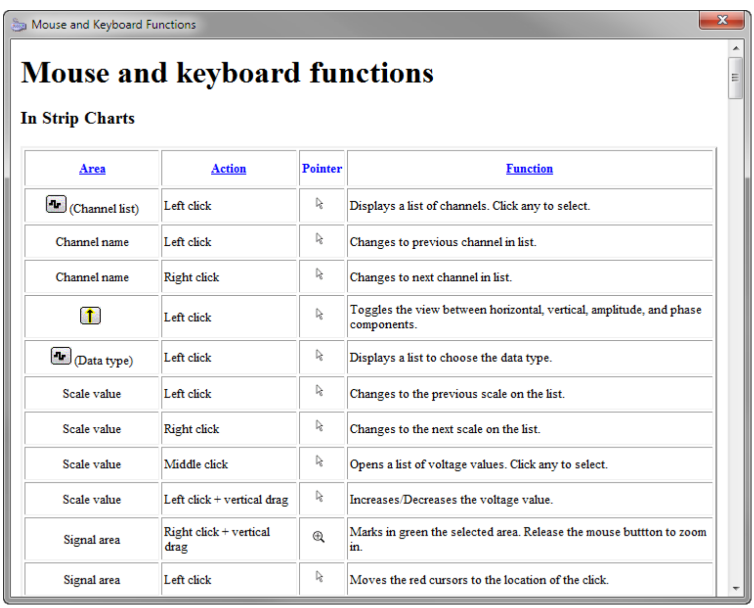
To do so, select **Mouse and Keyboard Functions** from the **Help** menu.

Figure 1-75 The **Help** menu



The **Mouse and Keyboard Functions** window appears.

Figure 1-76 The **Mouse and Keyboard Functions** window


Mouse and Keyboard Functions

Mouse and keyboard functions

In Strip Charts

Area	Action	Pointer	Function
(Channel list)	Left click		Displays a list of channels. Click any to select.
Channel name	Left click		Changes to previous channel in list.
Channel name	Right click		Changes to next channel in list.
	Left click		Toggles the view between horizontal, vertical, amplitude, and phase components.
(Data type)	Left click		Displays a list to choose the data type.
Scale value	Left click		Changes to the previous scale on the list.
Scale value	Right click		Changes to the next scale on the list.
Scale value	Middle click		Opens a list of voltage values. Click any to select.
Scale value	Left click + vertical drag		Increases/Decreases the voltage value.
Signal area	Right click + vertical drag		Marks in green the selected area. Release the mouse buttton to zoom in.
Signal area	Left click		Moves the red cursors to the location of the click.

For a complete list of mouse and keyboard functions, see “Mouse and Keyboard Functions” on page 325.

Exiting Magnifi

When you are done using Magnifi, you can exit the program in either of three ways:

- Select **Exit** from the **File** menu,
- Press ALT+F4 or,
- Click the close button in the upper right corner of the Magnifi main window.

Figure 1-77 The **File** menu

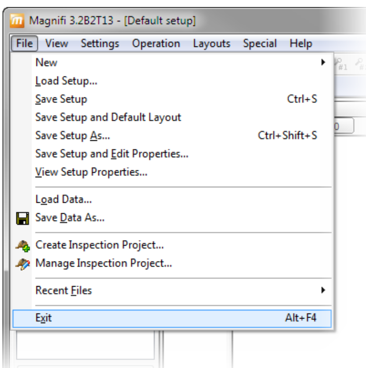
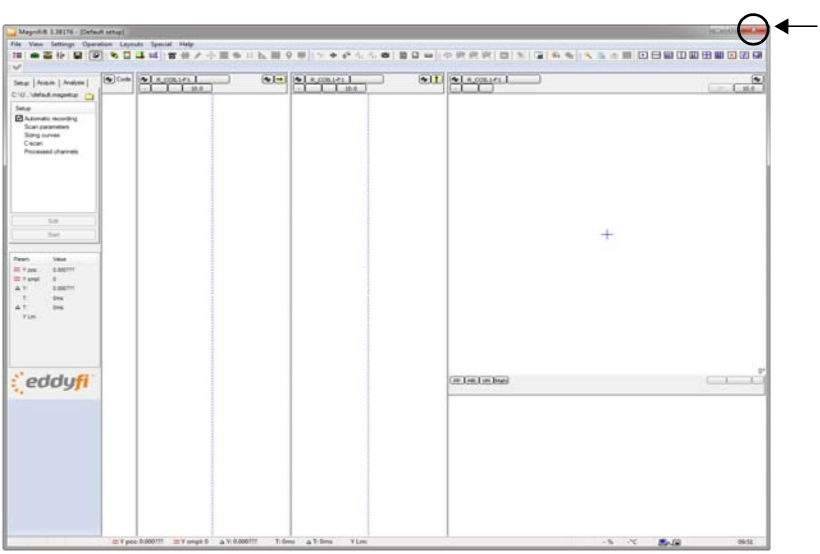


Figure 1-78 The Close button



Preparing Acquisition Setups with the Wizard

2

Setup Wizard

The setup wizard is a feature intended to simplify configuring the software for inspection. The wizard guides you through all the setup parameters necessary to properly and logically configure Magnifi for inspection purposes.

Component Geometries

The setup wizard is divided into two distinct categories of component geometries: tubes and surfaces. As understood by Magnifi, tubes are geometries that are typical in heat exchanger applications. These include internal and external probes that scan along the axis of tubes in a linear movement (360º coverage) or a helical movement (rotating probes). In both cases, conventional and array probes are supported.

Similarly, as understood by the software, surfaces are geometries that typically involve linear, raster, and polar scanning techniques on flat plates or large-diameter pipes. They include single-channel pencil probes and complex array probes.

Wizard Workflow

The wizard can accommodate approximately 85% of all possible inspection scenarios[1]. As such, parameters adapt at each step according to your selections. For example, the software only suggests probes adapted to the inspection geometry (tube from ID or surface) that you choose.

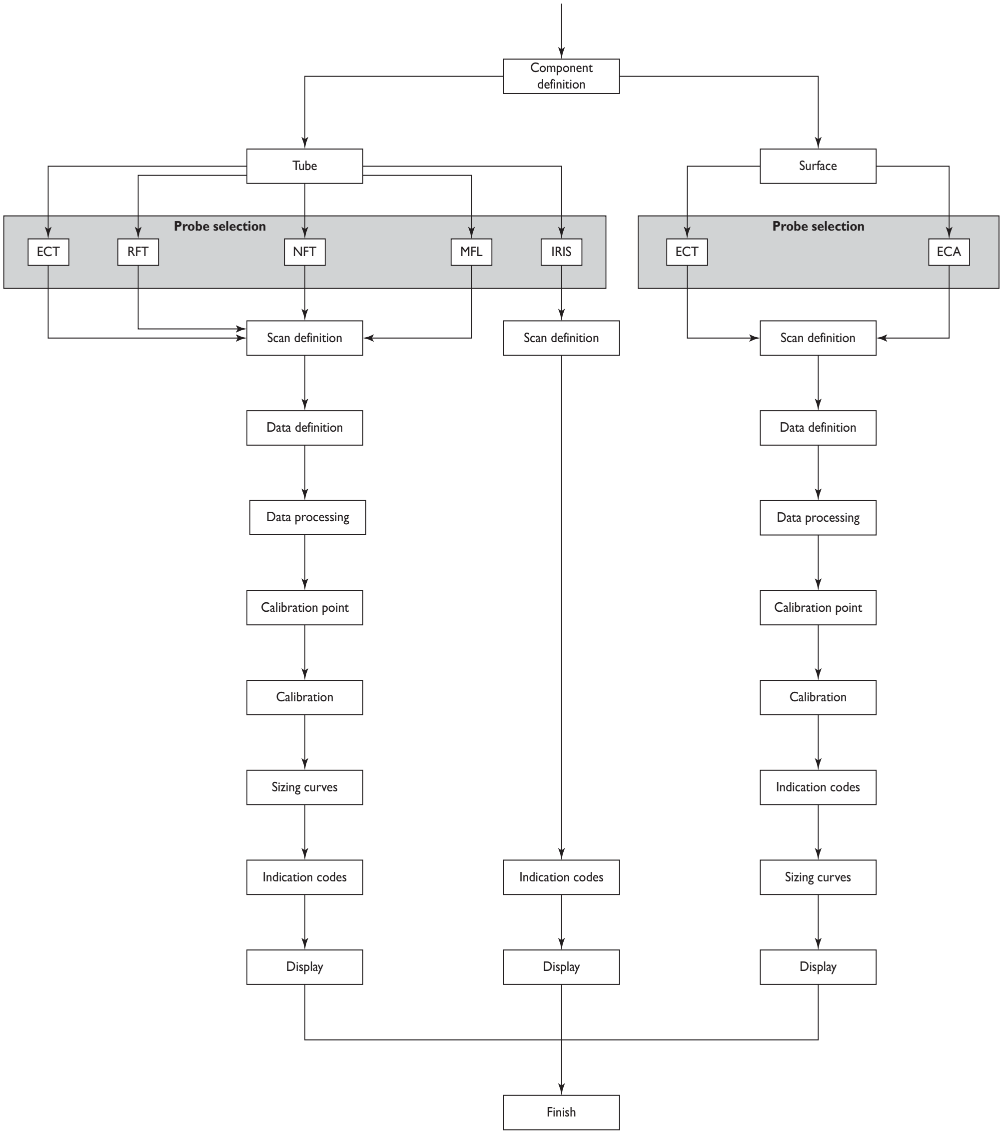
To start the wizard:

- Click **Settings>Setup Wizard>Create New Setup**
  or

- Click the **Setup Wizard** icon on the Setup Wizard toolbar.

1. For more customized scenarios, it is possible to configure the software manually, as explained later in this document (see “Preparing Conventional Acquisition Setups” on page 71).

Figure 2-1 Setup wizard workflow



- **Component definition —** Define the component that you need to test:
  - Geometry
  - Unit
  - Model type
  - Material
  - Dimensions
- **Probe selection —** Select the probe used for inspection. Use such filters as probe technique and model. After filtering, select one of the probes offered by Eddyfi from the list at the bottom.
- **Scan definition —** Define the parameters of the scan:
  - Scan type and axis
  - Start position obtained from
  - Acquisition rate
  - Typical probe speed
  - Typical axis parameters

  If you use an encoder, you will also be prompted to define its resolution.
- **Data definition —** Define the data used during inspection, as necessary:
  - Channels and/or C-scans
  - Frequencies
  - Mixes

  Also, select the Ectane connector used to acquire this data when the selection is possible.
- **Data processing —** Decide which filters — low-pass, high-pass — are applied to your working data.
- **Calibration point —** Define your calibration points:
  - Name
  - Side
  - Size
  - Units of measurement
- **Indication codes —** Define the indication codes necessary during inspection. By default, the wizard comes with a default list of indication codes, but you can modify the list and add to it as necessary.
- **Calibration —** Define the calibration characteristics of the channels that you are using
  - Amplitude and phase value
  - Reference
  - Measure

- **Sizing curves** — Create the sizing curves that you need:
  - Curve ID
  - Name
  - Source
  - Measure component
  - Method
  - Shape
  - etc.

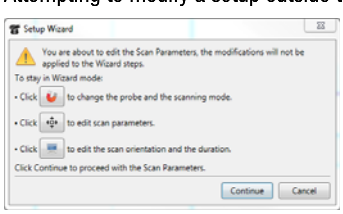
  You can add as many sizing curves as necessary. You are prompted to define the details of each curve, including the calibration points used to build the curve as you add them.
- **Data display:** Select how to display strip charts and C-scans, as well as the default layout that you want to use.

Interactions between the Setup Wizard and Magnifi

There are two ways of modifying a setup. With the wizard or through Magnifi menus. When you choose to modify a setup with the following Magnifi menus, you are prompted to return to the wizard:

- **Scan Parameters**
- **Sizing Curves**
- **C-Scan**
- **Processed Channels**
- **Instrument Configuration Wizard**

Figure 2-2 Attempting to modify a setup outside the wizard

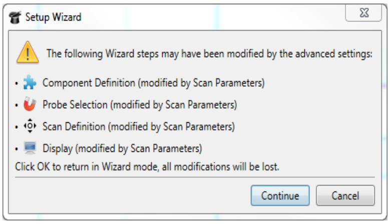


- To modify the setup outside the wizard, click **Continue**.

When you modify your setup without the wizard, the corresponding Magnifi dialog boxes appears, where you can make the necessary modifications to your setup. **Unless you save this modified setup under a different name, your modifications will be lost and will revert to the original configuration the next time you use it.**

If you attempt to modify a setup that was modified with regular Magnifi menus with the setup wizard, you are prompted as follows:

Figure 2-3 Attempting to modify a setup with the wizard



Clicking continue reverts your setup to its original wizard settings.

Preparing Conventional Acquisition Setups

3

Setting General Application Preferences

You must set certain application preferences before you start configuring any other part of the application. Measurement parameters are some of them.

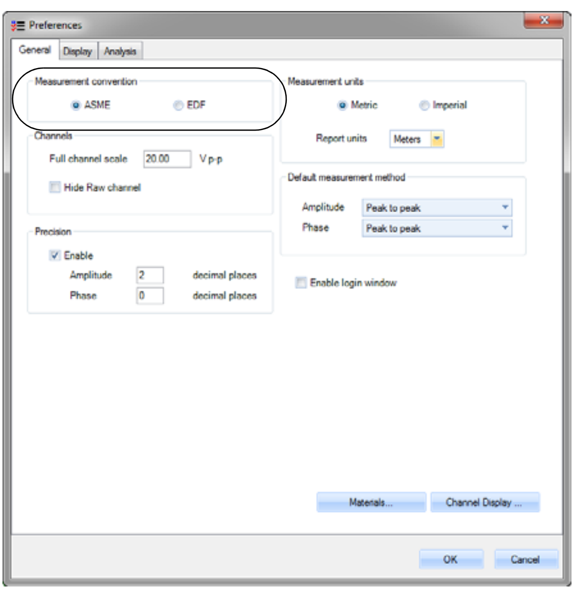
Setting Measurement Conventions

Magnifi allows you to choose between two different measurement conventions: ASME and EDF. The ASME (American Society of Mechanical Engineering) method is the most commonly used, whereas the EDF (*Électricité de France*) method is mostly used in France and within EDF subsidiaries (see Figure 3-2).

To choose a measurement convention:

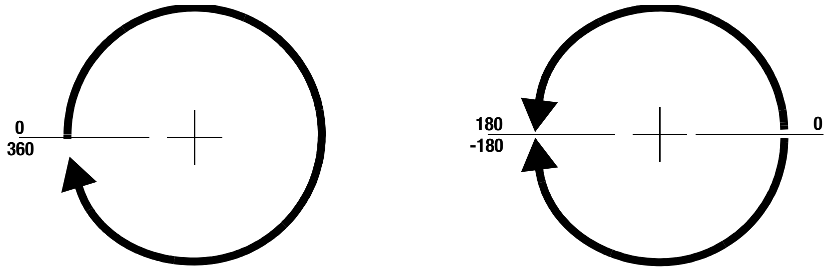
1. On the **Settings** menu, click **Preferences**. You can also click **Preferences** on the General toolbar. The **Preferences** window appears.

Figure 3-1 The **Preferences** window



2. In the **Measurement convention** section, select **ASME** (default) or **EDF.**

Figure 3-2 ASME (left) vs. EDF (right)



Phase is measured clockwise from 0° to 360°. The 0° is along the negative *x* axis.

Phase is measured counterclockwise from 0° to 180° starting along the positive *x* axis, and clockwise from 0° to -180° starting along the positive *x* axis.

3. Click **OK.** The measurement convention is set for the entire application.

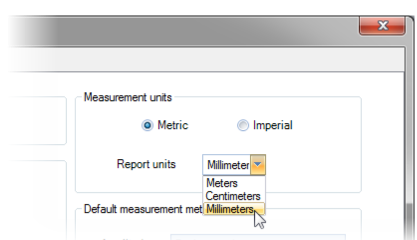
Setting Measurement Units

Magnifi allows you to choose between the metric and imperial measurement systems.

To do so:

1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. From the **Measurement units** section, select **Metric** or **Imperial.**
   - If you selected **Metric,** you can select the specific measurement unit used in reports from the **Report units** drop-down list (**m, cm,** or **mm**). If you selected **Imperial,** the only measurement unit offered is **in.**

Figure 3-3 The **Measurement units** section



3. Click **OK.** The **Preferences** window closes and the measurement units are set for the entire application.

Important Depending on the context elsewhere in the application, certain measurement units may alternate between the two measurement systems.

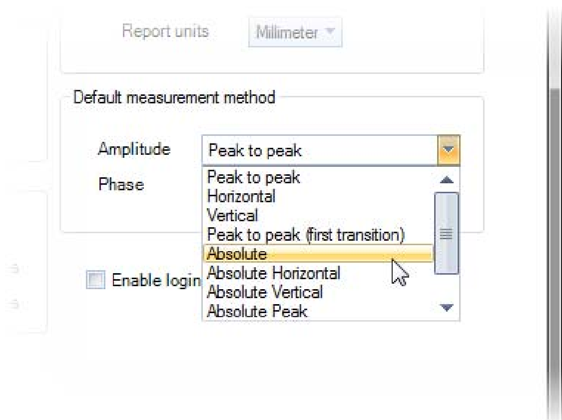
Setting Amplitude/Phase Measurement Methods

Magnifi allows you to set a default amplitude and/or phase measurement method for the entire application.

To do so:

1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. From the **Default measurement method** section of the **General** tab, select a default amplitude measurement method:

Figure 3-4 The **Default measurement method** section (amplitude)

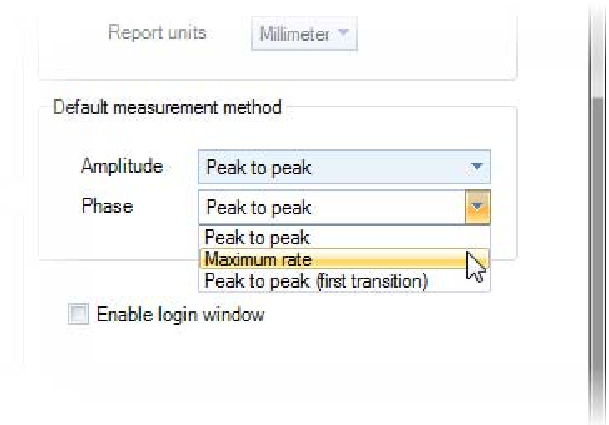


- **Peak to peak:** uses the combination of the vertical and horizontal component to measure the maximum amplitude.
- **Horizontal:** uses only the horizontal component to measure the amplitude.
- **Vertical:** uses only the vertical component to measure the amplitude.
- **Peak to peak (first transition):** uses the combination of the vertical and horizontal component of the first transition of the signal to measure maximum amplitude.
- **Absolute:** uses a straight line *from the zero position to the main cursor position* to measure amplitude.
- **Absolute Horizontal:** uses only the horizontal component, *from the imaginary zero level line to the main cursor position* to measure amplitude.
- **Absolute Vertical:** uses only the vertical component, *from the imaginary zero level line to the main cursor position* to measure amplitude.

- **Absolute Peak:** uses a straight line, *from the zero position to the peak value of the data selection,* to measure amplitude.
- **Absolute Peak Horizontal:** uses only the horizontal component, *from the imaginary zero position to the peak value of the data selection,* to measure amplitude.
- **Absolute Peak Vertical:** uses only the vertical component, *from the imaginary zero position to the peak value of the data selection,* to measure amplitude.

3. Select the default phase measurement method:

Figure 3-5 The **Default measurement method** section (phase)



- **Peak to peak**: measures the phase using the two points that are furthest apart in the Lissajous.
- **Maximum rate**: measures the phase using two points on the steepest section of the Lissajous.
- **Peak to peak (first transition):** measures the phase using the two points that are furthest apart on the first transition in the Lissajous. This method is particularly useful to measure absolute signal because the starting point of the measurement vector remains on the signal baseline.

4. Click **OK.** The **Preferences** window closes and the default amplitude and/or phase measurement methods are set for the entire application.

Important Depending on the context elsewhere in the application, other amplitude and/or phase measurement methods may be offered.

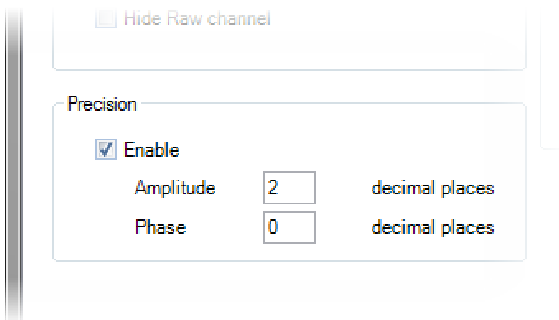
Setting Amplitude/Phase Measurement Precision

Magnifi allows you to display more precise amplitude and phase data by increasing the number of decimals shown in the interface.

To do so:

1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. In the **Precision** section, check the **Enable** box to activate the high-precision mode.

Figure 3-6 The **Precision** section



3. Enter the number of decimals that you need for amplitude and phase in their respective boxes.
4. Click **OK.** The **Preferences** window closes and the number of decimals entered for amplitude and phase will be used throughout Magnifi. Modifications will also be made in the appropriate Lissajous.

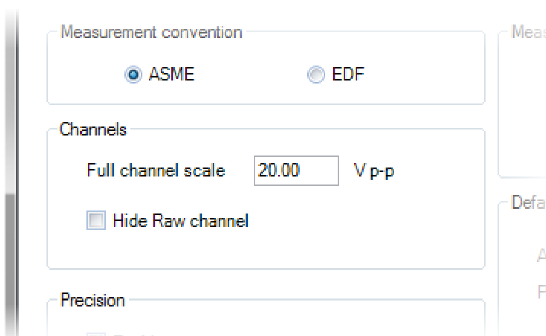
Setting the Full Channel Scale

Magnifi allows you to set the full scale for all channels. By default, the full channel scale is set at 20V p-p, but you can set a different scale. You can also hide the raw (unprocessed) channel.

To do so:

1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. In the **Channels** section of the **General** tab, enter the full scale that you want to use in the **Full channel scale** box.

Figure 3-7 The **Channels** section



3. To hide the raw (unprocessed) channels, check the **Hide Raw channel** box.
4. Click **OK.** The **Preferences** window closes and the channel scale that you defined will be used throughout Magnifi.

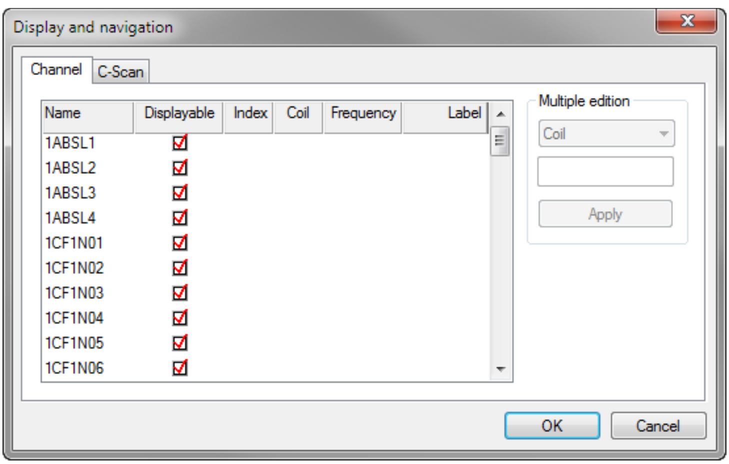
Setting Displayed Channel/C-scan Information

For each channel and C-scan, you can choose to assign and/or display certain information, namely an index and/or coil number, a frequency and a label.

To do so:

1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. Click **Channel Display.** in the lower right corner of the **General** tab. The **Display and navigation** window opens.

Figure 3-8 The **Display and navigation** window



3. On the **Channel** tab, you can assign an index and/or coil number, a frequency and a label to each available channel.
4. On the **C-scan** tab, you can assign the same information to each available C-scan.
5. To assign a single value to a series of coils, frequencies or labels, select the channels/C-scans to which you want to assign this value:
   - To select contiguous channels/C-scans in the list, click one channel/C-scan, press the SHIFT key, and click the last channel/C-scan that you want to select. All channels/C-scans between the first and last selected become selected as well.
   - To select non-contiguous channels/C-scans in the list, press the CTRL key while clicking on all the channels/C-scans that you want to select.

6. From the drop-down list in the **Multiple edition** section, select the element to which you want to assign a common value (**Coil, Frequency,** or **Label**).
7. In the text box underneath the drop-down list, enter the value to assign.
8. Click **Apply.** The value entered is applied to all selected channels/C-scans.
9. Click **OK** when you are done.

Setting Materials in Use

You can manage the materials in use from the **Material** window.

To do so:

1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. Click **Materials** in the lower right corner of the **General** tab. The **Materials** window opens.

For more information on working in the **Materials** window, see “Managing Materials” on page 186.

Managing Setup Properties

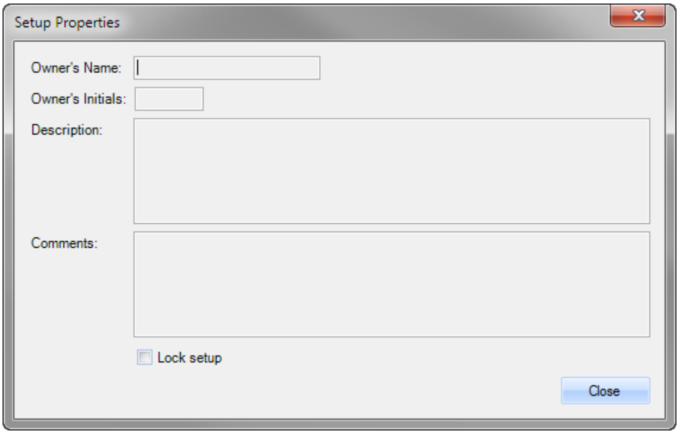
For each setup, you can view and edit certain properties.

Viewing Properties

To view properties:

1. From the **File** menu, select **View Setup Properties.** The **Setup Properties** window appears.

Figure 3-9 The **Setup Properties** window



2. From this window, you can see the name and the initials of the setup's owner as well as a description of the setup, and comments about it.

Editing Properties

To edit setup properties:

1. From the **File** menu, select **Save Setup and Edit Properties.** The **Setup Properties** window appears (see Figure 3-9). All fields shown in Figure 3-9 are editable.
2. Enter the relevant information in each field.
3. Click **OK.** The setup properties are saved.

Configuring Ectane ECT Acquisition Setups

Before configuring your scan parameters, you must configure the hardware that will use them. The following pages explain how to configure your hardware to perform acquisitions.

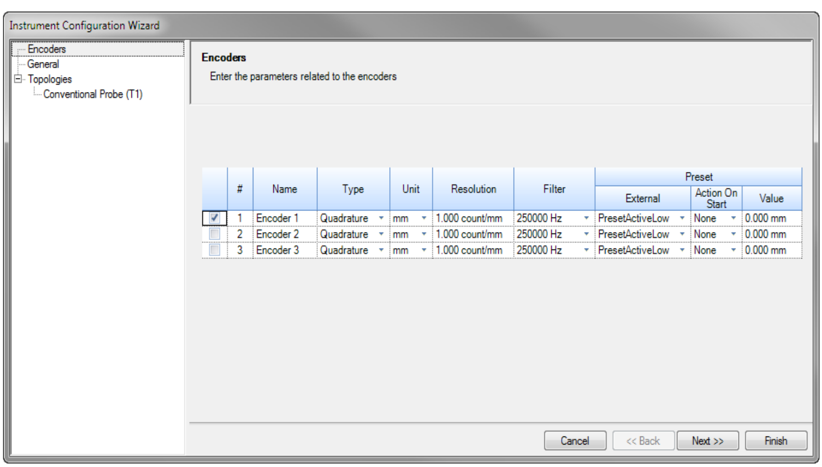
Important Magnifi also imports existing MS5800™ and TC7700™ setups. For more information on using these test units with Magnifi, see “Operating TC7700 & MS5800” on page 311.

Configuring an acquisition setup for the Ectane instrument is performed through a setup wizard that guides you, step by step, in entering all the relevant information regarding your setup. The following pages explain the various steps provided by the wizard.

To access the setup wizard, in the **Settings** menu, select **Acquisition Setup>Configuration Wizard.** The **Instrument Configuration Wizard** window opens. You can also click the **Configuration wizard** button in the Acquisition Setup toolbar.

Note *If the **Configuration Wizard** item is not available, first select **File>New> Ectane Setup**.*

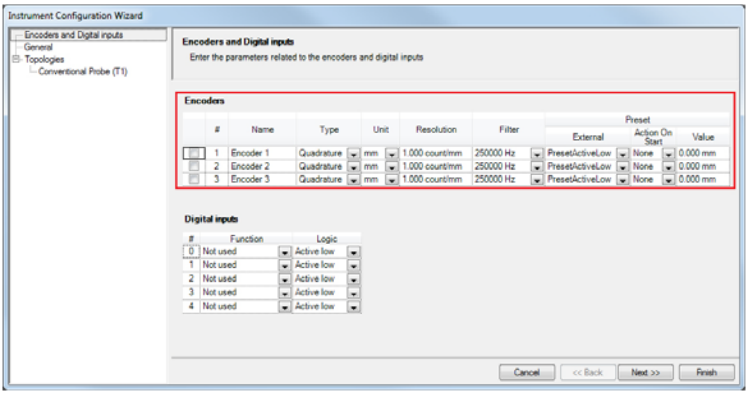
Figure 3-10 The **Instrument Configuration Wizard** window



Configuring Encoders and Digital Inputs

The first step offered by the configuration wizard, is configuration of the encoders and digital inputs, if any.

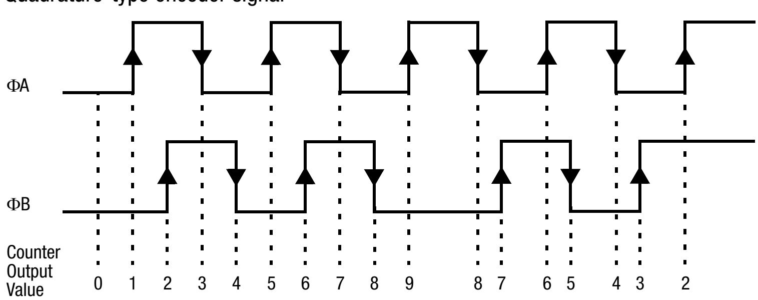
Figure 3-11 Encoders and Digital Inputs



To configure an encoder:

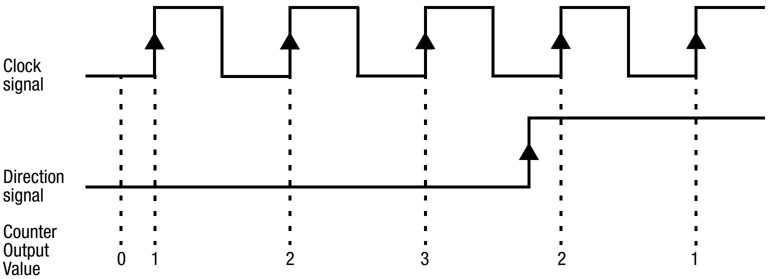
1. To activate that encoder configuration, select the check box before the line.
2. Click **Encoder *n*** under the **Name** column, and enter the name that you want to give to your encoder. Make sure that you give your encoder a name that will be meaningful to all users.
3. In the **Type** column, select the type of the encoder:
   - **Quadrature**: encoders producing two phase-quadrature signals, often called ΦA and ΦB signals. The quadrature counter is clocked by each transition of ΦA and ΦB signals, and the counting direction depends on the phase relationship of these signals. The counter counts up when ΦA leads ΦB; and it counts down when ΦA lags ΦB (see Figure 3-12).

Figure 3-12 Quadrature-type encoder signal



- **Clock dir**: encoders providing clock and direction signals, or clock signal only. The counter is clocked by each positive transition of the clock signal. The counting direction is controlled by the logic level of the direction signal. When this signal logic level is low, the counter counts up. When the logic level is high, the counter counts down (see Figure 3-13).

Figure 3-13 Clock dir encoder signal



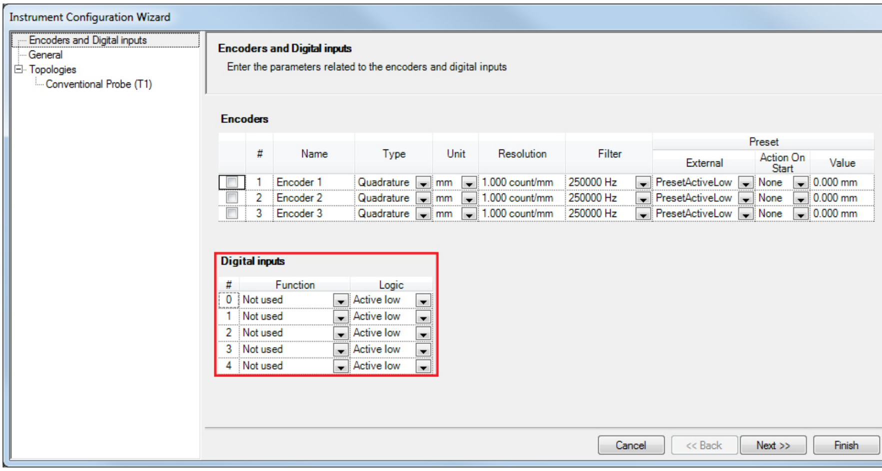
If the encoder provides a clock signal only, the counting direction may be software-controlled through the setup dialog box. In this case, the counter may be configured into UP or DOWN counter.

- **Up**: see explanation about Clock dir.
- **Down**: see explanation about Clock dir.

4. In the **Unit** column, select the appropriate measurement unit (**mm**, **in.**, **deg.**) The selected measurement unit has no impact on the application-wide measurement unit selected in the **General** tab of the **Preferences** window (see “Setting Measurement Units” on page 73).
5. Click in the **Resolution** field and enter the number of counts per unit of distance.
6. In the **Filter** column, select the low-pass filter that you want to apply to remove noise or to prevent triggering on glitches.
7. In the **External** sub-column of the **Preset** column, select what action must be performed by the encoder at the start of the acquisition:
   - **None**: no action is performed at the start of the acquisition.
   - **PresetActiveLow**: the acquisition starts when the logical level reaches 0V.
   - **PresetActiveHigh**: the acquisition starts when the logical level reaches 5V.
8. In the **Action On Start** sub-column, select whether (**Preset**) or not (**None**) an external source will be the trigger for the preset.
9. In the **Value** column, determine the value at which the preset will start.
10. Click **Next** to move to the next configuration step.

There are five available digital inputs.

Figure 3-14 Encoders and Digital Inputs

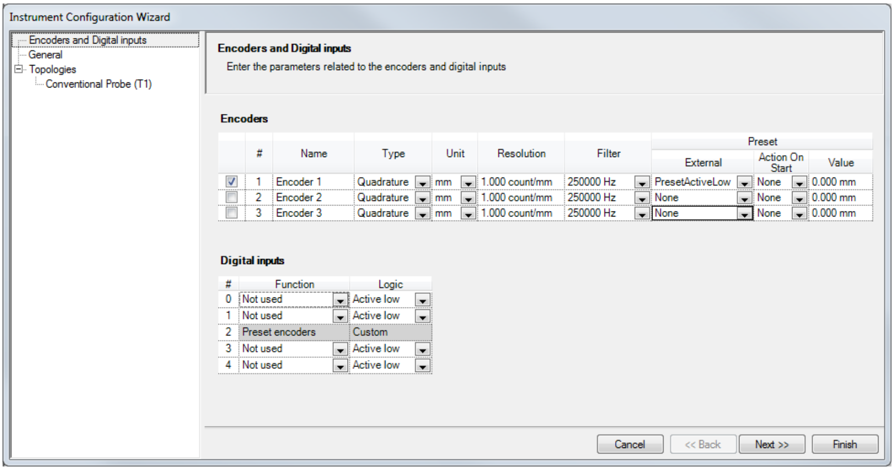


You can select how each input is used:

- When **Function** is **Not used**, the digital input signal is not considered by the Ectane.
- When **Function** is **Enable acquisition**. Only one of the inputs can be so configured. The function is used to stop data acquisition while maintaining the recorded file open. The function only works when the acquisition is based on an encoder. It can be used to momentarily pause the acquisition to move the scanner, for example.
- When **Function** is **Remote control**, the digital input is used to perform a remote software function based on active-low or active-high logic. See “Managing Remote Controls” on page 210.

Some digital inputs may not be available if encoder features are selected, because encoders and digital inputs share common pins on the connector, as illustrated here:

Figure 3-15 Inactive digital inputs according to encoder settings

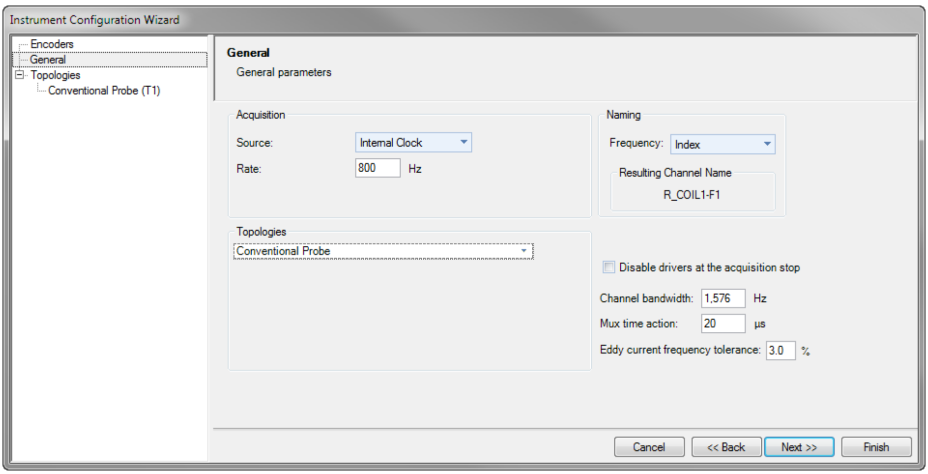


If you select an encoder and select an **External** preset, digital input 2 is dedicated to the preset function, for example.

Configuring General Acquisition Parameters

The second step offered by the configuration wizard is the configuration of general acquisition parameters.

Figure 3-16 Configuring general acquisition parameters



To configure general acquisition parameters:

1. In the **Acquisition** subsection, select the source of the acquisition rate from the **Source** drop-down list. From the drop-down list, you can select **Internal Clock, External Clock,** or any of the encoders defined in the previous step.
   - If you selected **Internal Clock,** enter the acquisition rate for that clock in the **Rate** field.
   - If you selected **External Clock,** enter the maximum acquisition rate for that clock in the **Maximum rate** field.
   - If you selected a predefined encoder, you have to select the direction of the scan from the **Direction** drop-down list (**Up and Down, Up, Down**), the maximum speed of the probe (in unit of distance per second) in the **Maximum probe speed** field, and the density of the acquisition (in number of samples per unit of distance) in the **Density** field.

     In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Applied** field.

2. In the **Naming** subsection, decide how raw channels will be named by selecting **Value** or **Index** from the **Frequency** drop-down list. The resulting channel naming convention is displayed in the **Resulting Channel Name** field.
3. If you want to deactivate the driver when there is no on-going acquisition, check the **Disable drivers at the acquisition step** box. This is useful to save batteries and to reduce heat on probes that require high drive voltage such as RFT probes.
4. Enter the bandwidth of every channel in the **Channel bandwidth** field.
5. Enter the time required by the multiplexer to stabilize its signal (per timeslot) in the **Mux time action** field.
6. Enter the percentage allowed of frequency variation in the **Eddy current frequency tolerance** field.
7. In the **Topologies** subsection, select the appropriate probe configuration:
   - **Conventional Probe** (default)
   - **Impedance**
   - **Double Driver**
   - **Short Double Driver**
   - **Orthogonal**
   - **Single Driver**
   - **DefHi Single Row**
   - **DefHi Double Row**
   - **Custom**

   The first topology is for conventional probe connections (4 pins, 19 pins, and 41 pins). All other topologies are for eddy current array probes with the SmartMUX™ (160-pin connector).
8. Click **Next** to move to the next configuration step.

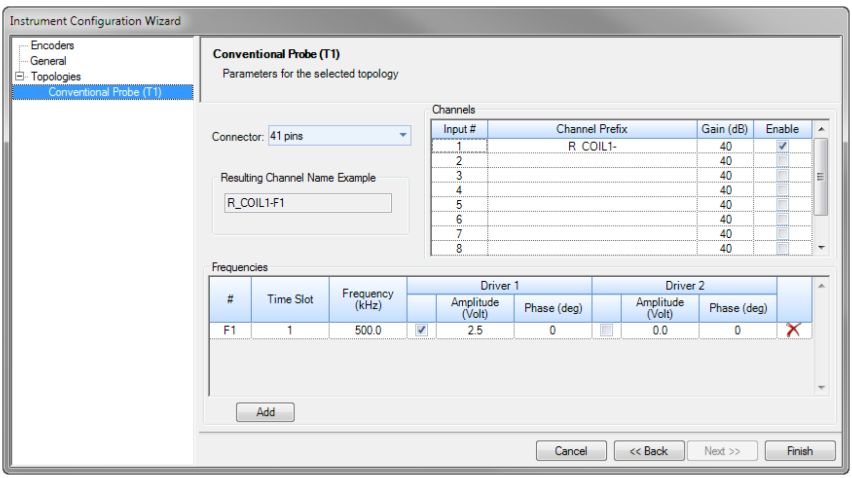
Configuring Probe Topologies

The last step offered by the configuration wizard is the configuration of the probe topology. The parameters to configure depend on the topology chosen in the previous step.

Configuring Conventional Topologies

To configure conventional topologies:

Figure 3-17 Conventional topology parameters



1. From the **Connector** drop-down list, select the Ectane connector that will be used: **4-pin, 19-pin, 19-pin MFL,** or **41-pin.**
2. In the **Channels** subsection, type a prefix in the **Channel Prefix** column.
3. The resulting channel name is displayed in the **Resulting Channel Name Example** box underneath the **Connector** drop-down list.

4. In the **Gain** column, enter the gain that you need. The proper value range depends on the selected connector:
   - **41-pin** and **4-pin** connectors: 23dB – 58dB
   - **19-pin:** 36dB – 86dB
   - **19-pin MFL:** 18dB – 53dB
5. In the **Enable** column, check the boxes next to the channels that you want to enable. Depending on the chosen connector, certain channels will already be disabled.

   If you chose:
   - **4-pin:** only inputs 5 and 6 will be available
   - **41-pin:** all inputs will be available
   - **19-pin** (**standard** or **MFL**): inputs 1 to 4 will be available.
6. By default, only one frequency is available in the **Frequencies** subsection (except if the 19-pin MFL connector is selected, see below).

   There are certain limitations pertaining to the **Frequencies** subsection:
   - The total amplitude for all frequencies cannot be more than 10V.
   - There cannot be more than five frequencies in use per timeslot.
   - There cannot be more timeslots than frequencies used.
   - When the 19-pin connector is used in MFL mode, frequencies are not available.

   a. Enter the timeslot number for each frequency on the **Time Slot** column. Frequencies in the same timeslot are generated simultaneously.
   b. Check the box of the driver that will generate the frequency.
   c. Enter the amplitude (in volts) and the phase (in degrees).

   You can add frequencies by clicking **Add**. When clicking **Add**, the limitations indicated above are automatically taken into consideration. Values are automatically recalculated to comply with the limitations.

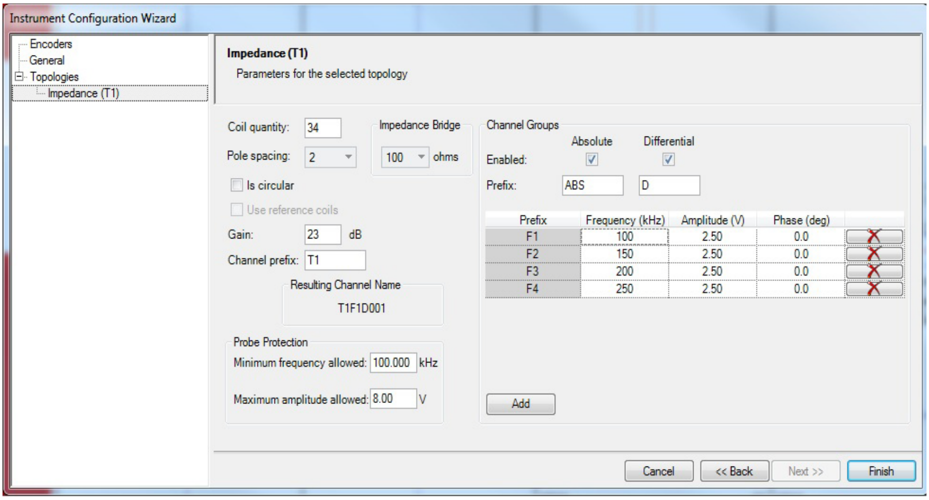
Note *You will get an error message if you edit the values manually to the point where you go over these limitations.*

7. Once you are done entering the topology window, click **Finish.** Your Ectane is now properly configured.

Configuring Impedance Topologies

To configure impedance topologies:

Figure 3-18 Impedance topology parameters

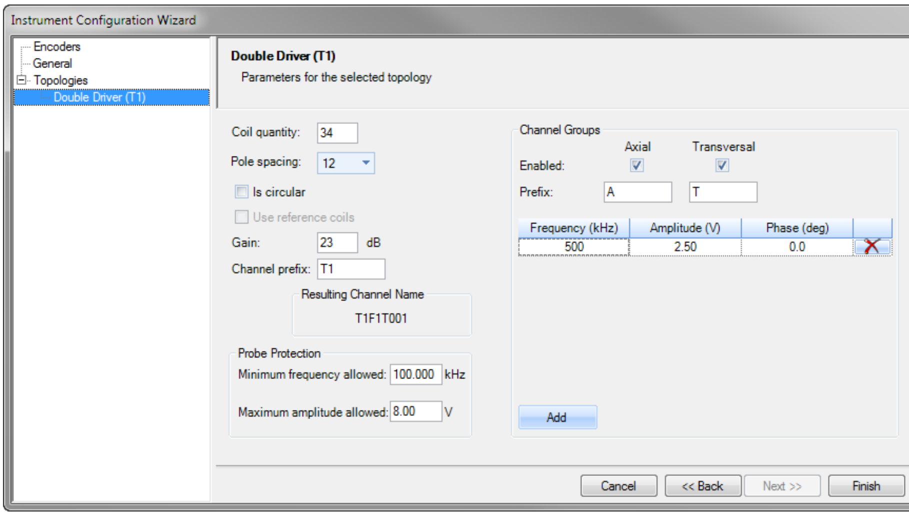


1. Enter the number of coils in your probe in the **Coil quantity** field. Spacing between poles is automatically set to **2** in the **Pole spacing** list.
2. Check the **Is circular** box if your impedance probe is circular.
3. Select the adequate impedance bridge value in the **Impedance Bridge** group.
4. Enter the required gain in the **Gain** field.
5. Enter a channel prefix in the **Channel prefix** field. The resulting channel name is indicated in the **Resulting Channel Name** field.
6. To properly protect the probe, enter the minimum frequency and maximum amplitude allowed in the **Minimum frequency allowed** and **Maximum amplitude allowed** fields of the **Probe Protection** subsection.
7. To enable channel groups, check the **Absolute** and/or **Differential** box(es) in the **Channel Groups** section. You can also assign prefixes to these channel groups in the **Prefix** fields under the appropriate column.
8. Enter the frequency, amplitude, and phase for each channel group that you need. To add frequencies, click **Add** at the bottom of the **Channel Groups** section. This adds a new line of data (maximum of four) under the first, where you can also enter the proper frequency, amplitude, and phase.
9. When you are done configuring the topology window, click **Finish.** Your Ectane is now properly configured.

Configuring Double Driver Topologies

To configure double driver topologies:

Figure 3-19 Double driver topology parameters

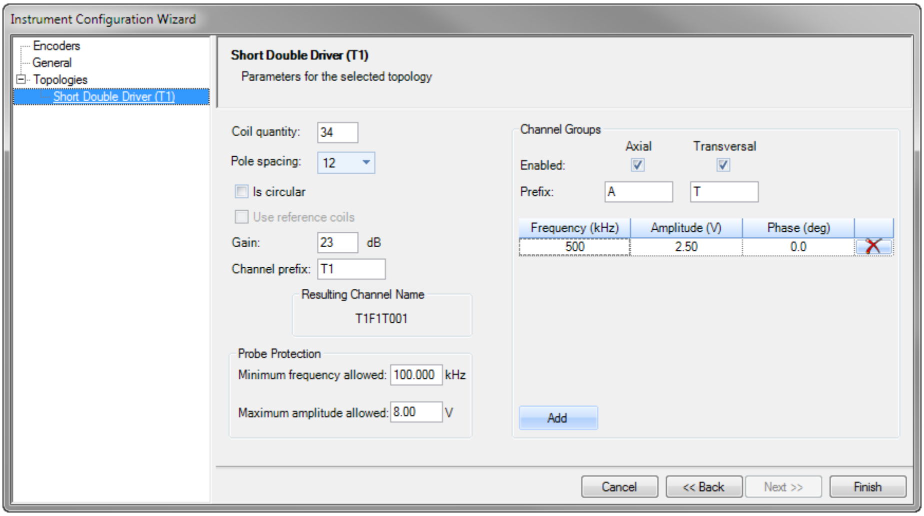


1. Enter the number of coils in your probe in the **Coil quantity** field.
2. Select the required spacing between poles in the **Pole spacing** drop-down list.
3. Check the **Is circular** box if your double driver probe is circular.
4. Enter the required gain in the **Gain** field.
5. Enter a channel prefix in the **Channel prefix** field. The resulting channel name is indicated in the **Resulting Channel Name** field.
6. To properly protect the probe, enter the minimum frequency and maximum amplitude allowed in the **Minimum frequency allowed** and **Maximum amplitude allowed** fields of the **Probe Protection** section.
7. To enable channel groups, check the **Axial** and/or **Transversal** box(es) in the **Channel Groups** section. You can also assign prefixes to these channel groups in the **Prefix** fields under the appropriate column.
8. Enter the frequency, amplitude, and phase for each channel group that you need. To add frequencies, click **Add** at the bottom of the **Channel Groups** section. This adds a new line of data (maximum of four) under the first, where you can also enter the proper frequency, amplitude, and phase.
9. Once you are done entering the topology window, click **Finish.** Your Ectane is now properly configured.

Configuring Short Double Driver Topologies

To configure short double driver topologies:

Figure 3-20 Short double driver topology parameters

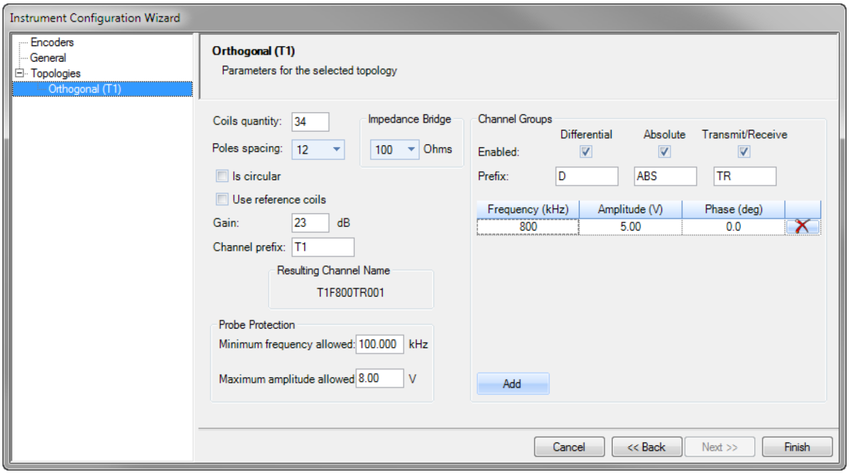


1. Enter the number of coils in your probe in the **Coil quantity** field.
2. Select the required spacing between poles in the **Pole spacing** drop-down list.
3. Check the **Is circular** box if your short double driver probe is circular.
4. Enter the required gain in the **Gain** field.
5. Enter a channel prefix in the **Channel prefix** field. The resulting channel name is indicated in the **Resulting Channel Name** field.
6. To properly protect the probe, enter the minimum frequency and maximum amplitude allowed in the **Minimum frequency allowed** and **Maximum amplitude allowed** fields of the **Probe Protection** section.
7. To enable channel groups, check the **Axial** and/or **Transversal** box(es) in the **Channel Groups** section. You can also assign prefixes to these channel groups in the **Prefix** fields under the appropriate column.
8. Enter the frequency, amplitude, and phase for each channel group that you need. To add frequencies, click **Add** at the bottom of the **Channel Groups** section. This adds a new line of data (maximum of four) under the first, where you can also enter the proper frequency, amplitude, and phase.
9. Once you are done entering the topology window, click **Finish.** Your Ectane is now properly configured.

Configuring Orthogonal Topologies

To configure orthogonal topologies:

Figure 3-21 Orthogonal topology parameters

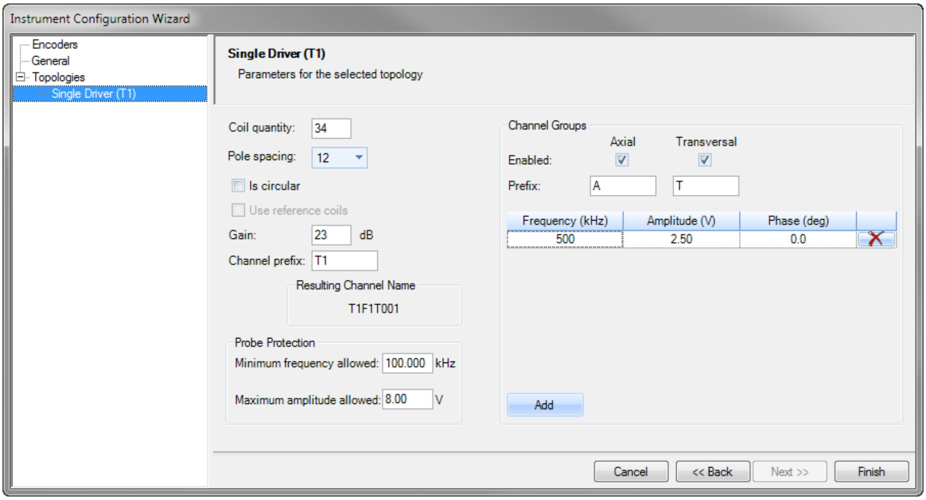


1. Enter the number of coils in your probe in the **Coil quantity** field.
2. Select the required spacing between poles in the **Pole spacing** drop-down list.
3. Select the proper impedance bridge from the drop-down list in the **Impedance Bridge** subsection.
4. Check the **Is circular** box if your orthogonal probe is circular.
5. Enter the required gain in the **Gain** field.
6. Enter a channel prefix in the **Channel prefix** field. The resulting channel name is indicated in the **Resulting Channel Name** field.
7. To properly protect the probe, enter the minimum frequency and maximum amplitude allowed in the **Minimum frequency allowed** and **Maximum amplitude allowed** fields of the **Probe Protection** section.
8. To enable channel groups, check the **Differential, Absolute,** and/or **Transmit/Receive** box(es) in the **Channel Groups** section. You can also assign prefixes to these channel groups in the **Prefix** fields under the appropriate column.
9. Enter the frequency, amplitude, and phase for each channel group that you need. To add channel groups, click **Add** at the bottom of the **Channel Groups** section. This adds a new line of data (maximum of four) under the first, where you can also enter the proper frequency, amplitude, and phase.
10. Once you are done entering the topology window, click **Finish.** Your Ectane is now properly configured.

Configuring Single Driver Topologies

To configure single driver topologies:

Figure 3-22 Single driver topology parameters

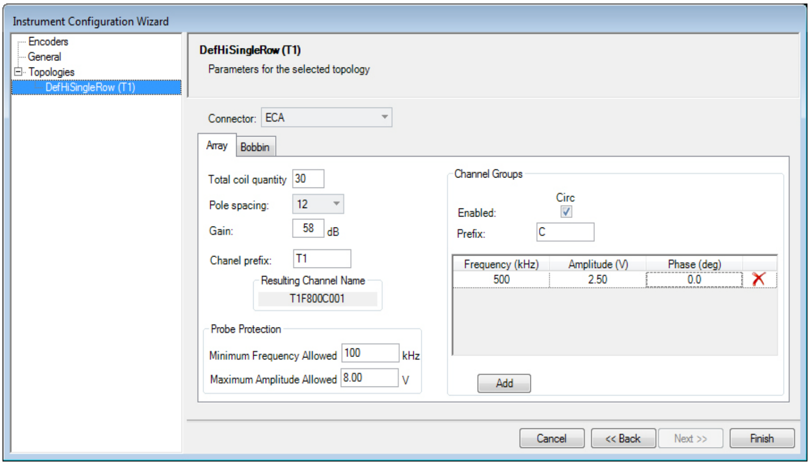


1. Enter the number of coils in your probe in the **Coil quantity** field.
2. Select the required spacing between poles in the **Pole spacing** drop-down list.
3. Check the **Is circular** box if your X-probe is circular.
4. Enter the required gain in the **Gain** field.
5. Enter a channel prefix in the **Channel prefix** field. The resulting channel name is indicated in the **Resulting Channel Name** field.
6. To properly protect the probe, enter the minimum frequency and maximum amplitude allowed in the **Minimum frequency allowed** and **Maximum amplitude allowed** fields of the **Probe Protection** section.
7. To enable channel groups, check the **Axial** and/or **Transversal** box(es) in the **Channel Groups** section. You can also assign prefixes to these channel groups in the **Prefix** fields under the appropriate column.
8. Enter the frequency, amplitude, and phase for each channel group that you need. To add frequencies, click **Add** at the bottom of the **Channel Groups** section. This adds a new data line (maximum of five) under the first, where you can also enter the proper frequency, amplitude, and phase.
9. Once you are done entering the topology window, click **Finish.** Your Ectane is now properly configured.

Configuring DefHi Single-Row Topologies

To configure DefHi single-row topology array parameters:

Figure 3-23 DefHi single-row topology array parameters



1. In the **Total coil quantity** text box, type the number of coils in your probe. The **Pole spacing** list is always configured to **12** for DefHi probes.
2. In the **Gain** text box, type the required gain.
3. In the **Channel prefix** text box, type a channel prefix.
   The resulting channel name is indicated in the **Resulting Channel Name** box.
4. To properly protect the probe, specify the allowed minimum frequency and maximum amplitude in the **Minimum frequency allowed** and **Maximum amplitude allowed** text boxes of the **Probe Protection** group.
5. To enable channel groups, select the **Circ** check box in the **Channel Groups** group.
   You can also assign prefixes to channel groups in the **Prefix** text box in the appropriate column.
6. Type the frequency (in kHz), the amplitude (in V), and the phase (in degrees) that you need for the Circ group.
   a To add frequencies, click **Add** at the bottom of the **Channel Groups** group. This adds a new data line (maximum of five) under the first, where you can also enter the proper frequency, amplitude, and phase.

To configure DefHi single-row topology bobbin parameters:

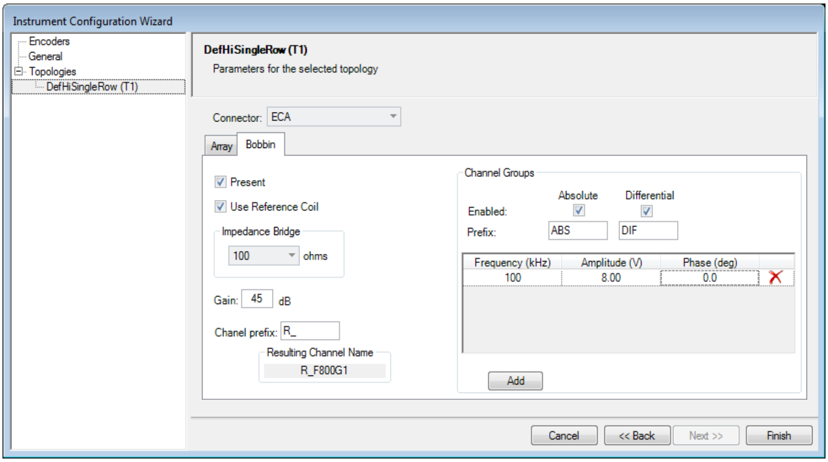


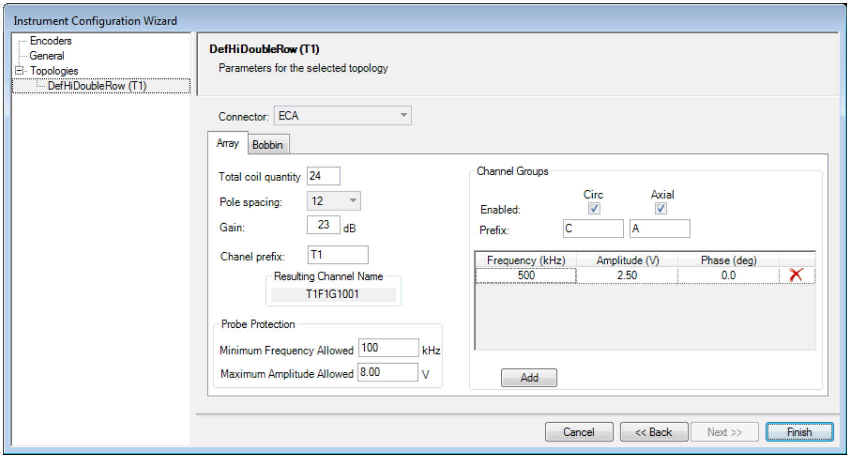
Figure 3-24 DefHi topology bobbin parameters

1. To enable the bobbin portion of your DefHi probe, select the **Present** check box.
2. If your DefHi probe contains reference coils, select the **Use reference coils** check box.
3. On the **Impedance Bridge** group's list, select the appropriate bridge impedance.
4. In the **Gain** text box, type an appropriate value.
5. In the **Channel prefix** text box, type a channel prefix.
   The resulting channel name is displayed in the **Resulting Channel Name** box.
6. To enable channel groups, select the **Absolute** and/or **Differential** check box in the **Channel Groups** group.
   You can also assign prefixes to these channel groups in the **Prefix** box in the appropriate column.
7. Specify the frequency (in kHz), the amplitude (in V), and the phase (in degrees) for each necessary group of channels.
   a To add frequencies, click **Add** at the bottom of the **Channel Groups** group. This adds a new data line (maximum of five) under the first, where you can also enter an appropriate frequency, amplitude, and phase.
8. When you have finished configuring the topology, click **Finish**.
   Your Ectane is now properly configured.

Configuring DefHi Double-Row Topologies

To configure DefHi double-row topology array parameters:

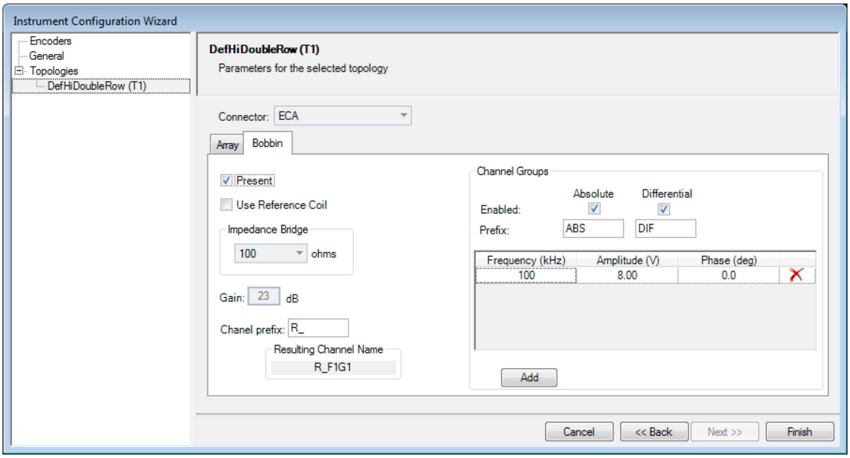
Figure 3-25 DefHi double-row topology array parameters



1. In the **Total coil quantity** text box, type the number of coils in your probe. The **Pole spacing** list is always configured to **12** for DefHi probes.
2. In the **Gain** text box, type the required gain.
3. In the **Channel prefix** text box, type a channel prefix. The resulting channel name is indicated in the **Resulting Channel Name** box.
4. To properly protect the probe, specify the allowed minimum frequency and maximum amplitude in the **Minimum frequency allowed** and **Maximum amplitude allowed** text boxes of the **Probe Protection** group.
5. To enable channel groups, select the **Circ** and/or **Axial** check box in the **Channel Groups** group.
   You can also assign prefixes to channel groups in the **Prefix** text box in the appropriate column.
6. Type the frequency (in kHz), the amplitude (in V), and the phase (in degrees) that you need for the Circ group.
   a To add frequencies, click **Add** at the bottom of the **Channel Groups** group. This adds a new data line (maximum of five) under the first, where you can also enter the proper frequency, amplitude, and phase.

To configure DefHi double-row topology bobbin parameters:

Figure 3-26 DefHi topology bobbin parameters



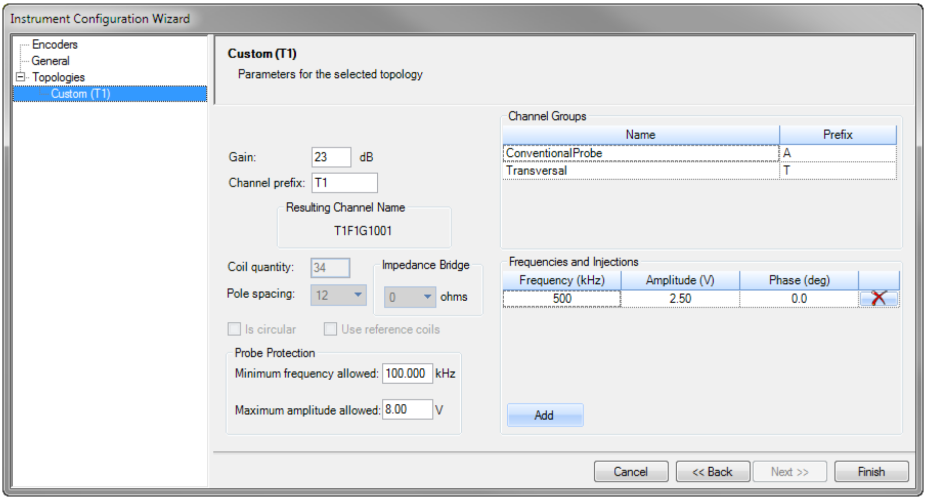
1. To enable the bobbin portion of your DefHi probe, select the **Present** check box.
2. If your DefHi probe contains reference coils, select the **Use reference coils** check box.
3. On the **Impedance Bridge** group's list, select the appropriate bridge impedance.
4. In the **Gain** text box, type an appropriate value.
5. In the **Channel prefix** text box, type a channel prefix.
   The resulting channel name is displayed in the **Resulting Channel Name** box.
6. To enable channel groups, select the **Absolute** and/or **Differential** check box in the **Channel Groups** group.
   You can also assign prefixes to these channel groups in the **Prefix** box in the appropriate column.
7. Specify the frequency (in kHz), the amplitude (in V), and the phase (in degrees) for each necessary group of channels.
   a To add frequencies, click **Add** at the bottom of the **Channel Groups** group.
   This adds a new data line (maximum of five) under the first, where you can also enter an appropriate frequency, amplitude, and phase.
8. When you have finished configuring the topology, click **Finish**.
   Your Ectane is now properly configured.

Configuring Custom Topologies

To configure a custom probe, you need the PDK and a solid knowledge of probe design.

To configure custom topologies:

Figure 3-27 Custom topology parameters



1. Enter the required gain in the **Gain** field.
2. Enter a channel prefix in the **Channel prefix** field. The resulting channel name is indicated in the **Resulting Channel Name** field.
3. Enter the number of coils in your probe in the **Coil quantity** field.
4. Select the required spacing between poles in the **Pole spacing** drop-down list.
5. Select the proper impedance bridge from the drop-down list in the **Impedance Bridge** subsection.
6. Check the **Is circular** box if your probe is circular.
7. Check the **Use reference coils** box if your probe contains reference coils
8. To properly protect the probe, enter the minimum frequency and maximum amplitude allowed in the **Minimum frequency allowed** and **Maximum amplitude allowed** fields of the **Probe Protection** section.
9. Enter the name and prefix of the channel group in the **Name** and **Prefix** column of the **Channel Groups** section.
10. Enter the frequency, amplitude, and phase of your probe in the **Frequencies and Injections** section.
11. Once you are done entering the topology window, click **Finish.** Your Ectane is now properly configured.

Configuring Ectane IRIS Acquisition Setups

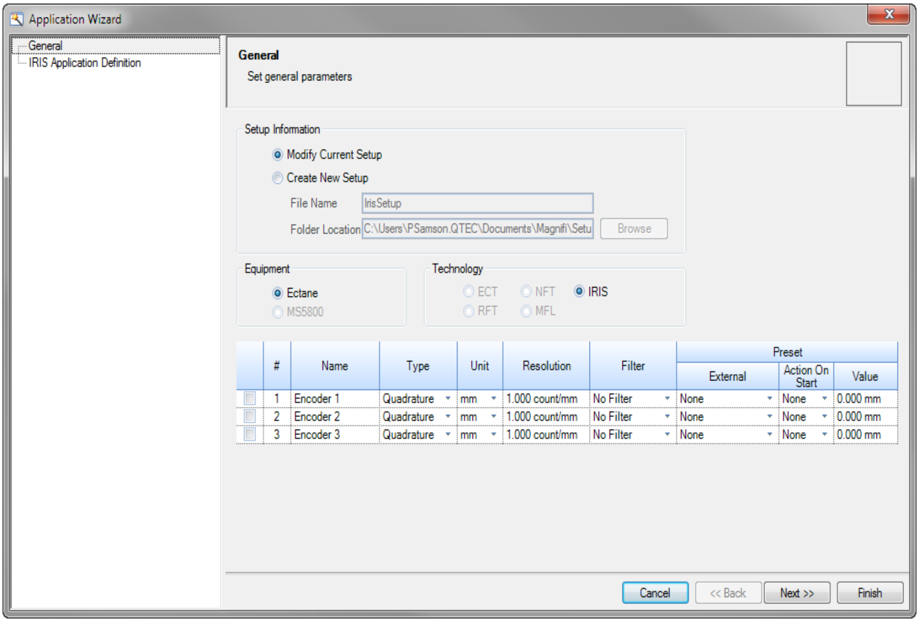
Before configuring your scan parameters for use with IRIS probes, you must configure the hardware that will use them. As of this writing, Magnifi only supports acquisitions via Eddyfi's Ectane instrument.

Note *Magnifi can neither work with existing MS5800™ IRIS setups, nor convert them for use with the Ectane.*

The following pages explain how to configure your hardware to perform IRIS acquisitions. Configuring an acquisition setup for the Ectane instrument is performed through the Application Wizard, that guides you, step by step, in entering all the relevant information regarding your setup. The following pages explain the various steps provided by the wizard.

To access the Application Wizard, in the **Settings** menu, select **Application Wizard.** The **Application Wizard** window opens.

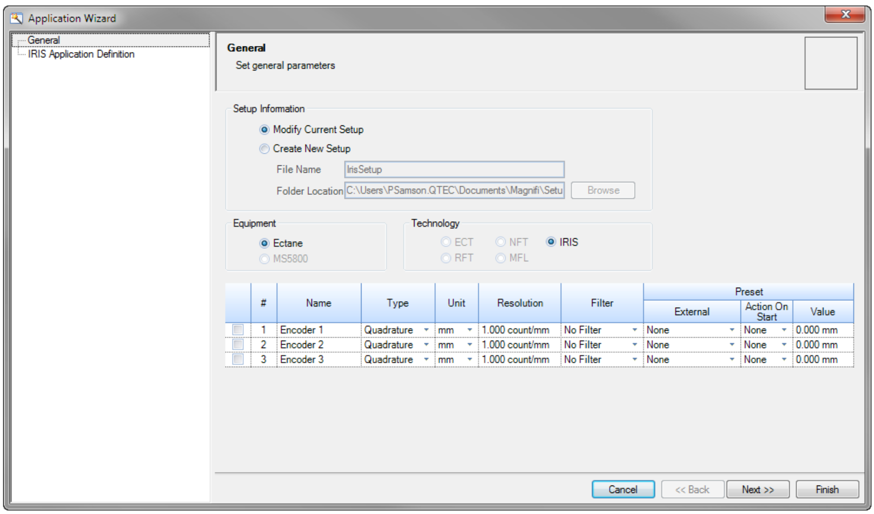
Figure 3-28 The Application Wizard window



Configuring General IRIS Parameters

When the **Application Wizard** appears, the **General** page is selected. From there, you select an inspection setup, the equipment used (test instrument and encoders) as well as the technology used (only IRIS is available in version 3.3).

Figure 3-29 The **General** page of the **Application Wizard** window



To do so:

1. In the **Setup Information** section, select whether you want to modify the setup currently loaded (default), or create a new one.

Note *The **Modify Current Setup** option is only available if the current setup is compatible with IRIS inspections.*

   a. To create a new setup, select **Create New Setup.** The **File Name** and **Folder Location** text fields become active.
   b. Enter the name of the setup in the **File Name** field.
   c. Select the location where you want to save this setup by clicking **Browse** and then finding that location (the location selected by default is the folder where all other predefined setups are saved).

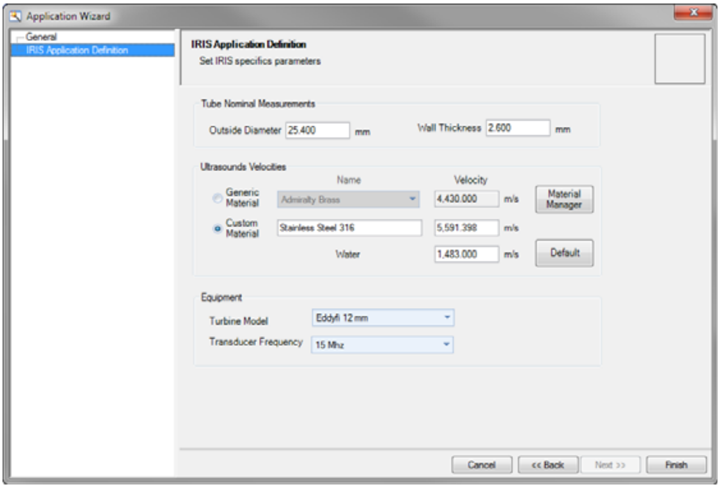
2. In the **Equipment** section, select the instrument that you will use for your IRIS inspection (in version 3.3 of the software, only the Ectane is available).
3. In the **Technology** section, select the inspection technology that you want to use (in version 3.3 of the software, only IRIS is available).
4. Check the box at the beginning of the encoder line to activate that encoder configuration.

5. Click **Encoder *n*** under the **Name** column, and enter the name that you want to give to your encoder. Make sure that you give your encoder a name that will be meaningful to all users.
6. In the **Type** column, select the type of the encoder. For more information on the types of encoders, see “Configuring Encoders and Digital Inputs” on page 82.
7. In the **Unit** column, select the appropriate measurement unit (**mm**, **in.**, **deg.**) The selected measurement unit has no impact on the application-wide measurement unit selected in the **General** tab of the **Preferences** window (see “Setting Measurement Units” on page 73).
8. Click in the **Resolution** field and enter the number of counts per unit of distance.
9. In the **Filter** column, select the low-pass filter that you want to apply to remove noise or to prevent triggering on glitches.
10. In the **External** sub-column from the **Preset** column, select what action must be performed by the encoder at the start of the acquisition:
    - **None**: no action is performed at the start of the acquisition.
    - **PresetActiveLow**: the acquisition starts when the logical level reaches 0V.
    - **PresetActiveHigh**: the acquisition starts when the logical level reaches 5V.
11. In the **Action On Start** sub-column, select whether (**Preset**) or not (**None**) an external source will be the trigger for the preset.
12. In the **Value** column, determine the value at which the preset will start.
13. Click **Next>>** to move to the next configuration step.

Configuring IRIS-specific Parameters

Once you have clicked **Next,** you are brought to the **IRIS Application Definition** page.

Figure 3-30 The **IRIS Application Definition** page



1. In the **Tube Nominal Measurements** section, enter the tube outside diameter in the **Outside Diameter** text box, and the tube wall thickness in the **Wall Thickness** text box.
2. In the **Ultrasound Velocities** section, select the material in which the ultrasound waves will be propagated. You can select either a standard material (**Generic Material**) or a new material (**Custom Material**):
   a If you selected a generic material, the drop-down list under the **Name** column becomes active and you can select one of the listed materials. The remaining fields are automatically completed.
   b If you selected a custom material, enter the name of the material in the **Name** column text box, and the velocity of sound in it in the **Velocity** text box.
   c You also have the velocity of sound in water, which you can change if necessary (if the water used has been modified in some way which would have changed its behavior in relation to how sound travels in it.) (Click **Default** to return the default water velocity value).
3. In the **Equipment** section, select the turbine used in the **Turbine Model** drop-down list.
4. In the same section, select the transducer frequency in the **Transducer Frequency** drop-down list.
5. Click **Finish** to finish your IRIS configuration.

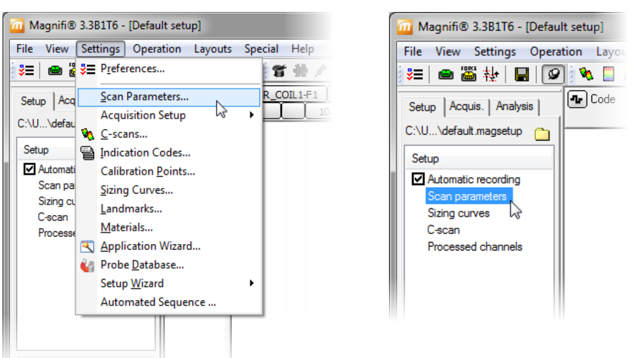
Configuring Scan Parameters

When preparing an acquisition setup, the scan parameters to set depend on the type of scan that you need to perform. Here is a list of scan types supported by Magnifi:

- Linear, single axis (with bobbin probes)
- Tube/bore
  - with rotating probes (**note:** *also used when performing IRIS inspections*)
  - with array probes
  - with rotating array probes
- 2D surface
  - raster, single channel
  - single pass array probe
  - raster, array probe
  - polar array
  - polar raster

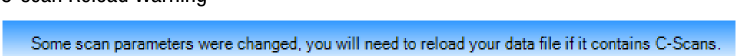
Scan parameters are set from the **Scan Parameters** window that appears when selecting **Scan Parameters** from the **Settings** menu, or when double-clicking **Scan parameters** in the **Setup** tab.

Figure 3-31 Accessing the scan parameters



Important C-scans are not automatically updated when modifying scan parameters. Whenever you make changes that might affect the display of C-scans, the following message will pop up. Simply click the message to make it go away.

Figure 3-32 C-scan Reload Warning

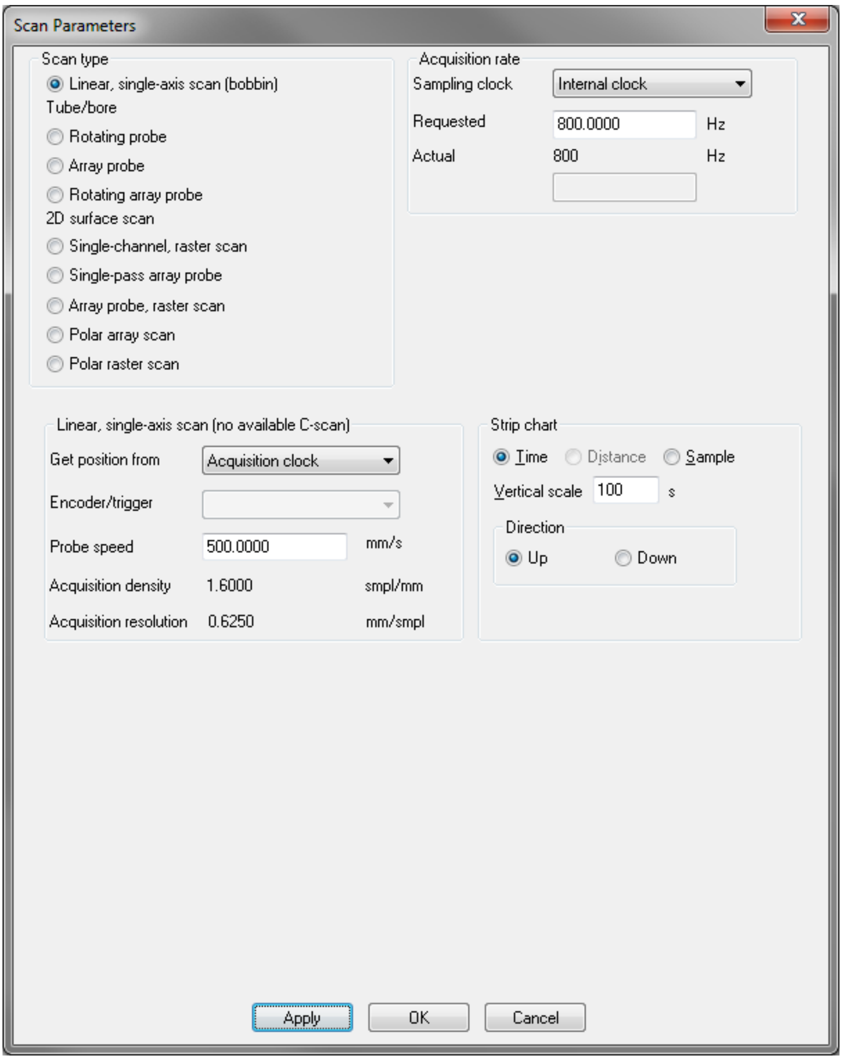


For Linear Single-Axis Scans

To set up scan parameters for linear single-axis scans (i.e., with bobbin probes):

1. Select **Scan Parameters** from the **Settings** menu or double-click **Scan parameters** in the **Setup** tab. The **Scan Parameters** window appears.

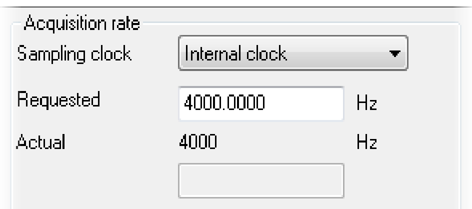
Figure 3-33 The **Scan Parameters** window for linear single-axis scans



2. In the **Scan type** section, select **Linear, single-axis scan (bobbin)**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder).

Note *The encoder option is available only if an encoder has been previously defined. For more information on defining encoders, see page 82.*

Figure 3-34 The **Acquisition rate** section: **Internal clock** selected



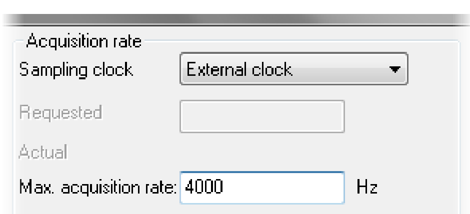
a From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.

b If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.

In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.

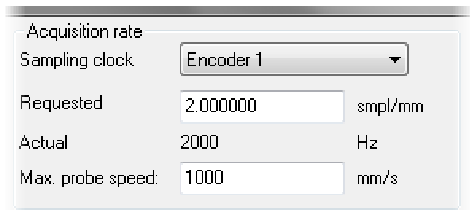
c If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.

Figure 3-35 The **Acquisition rate** section: **External clock** selected



d If you chose a preset encoder, you have to enter the requested resolution in the **Requested** field, and the maximum probe speed in the **Max. probe speed** field.

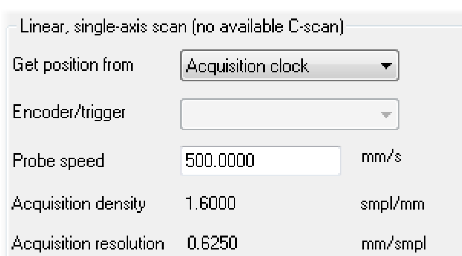
Figure 3-36 The **Acquisition rate** section: **Encoder** selected



Note *Depending on how the encoder was defined, acquisition rate measurement units might differ from the ones selected for the application in the* ***General*** *tab of the* ***Preferences*** *dialog box (see "Setting Measurement Units" on page 73).*

4. In the **Linear, single-axis scan (no available C-scan)** section, configure your basic scan parameters:

Figure 3-37 The **Linear, single-axis scan (no available C-scan)** section



a In the **Get position from** drop-down list, select what device will provide probe positions. The devices shown in the list depend on the selection made in step 3.

If you selected **Internal clock** in step 3, you can choose from **Acquisition clock** (default value), **Non-standard clock,** or **Encoder.**

If you selected **External clock** or a preset encoder in step 3, you can only select **Encoder.**

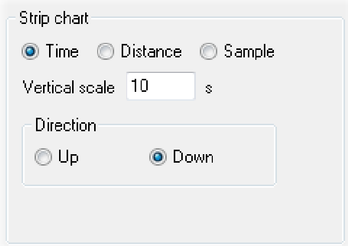
b When selecting **Encoder** from the **Get position from** drop-down list, you must select the proper encoder in the **Encoder/trigger** drop-down list.

c Enter the estimated probe speed in the **Probe speed** field. If you selected **Acquisition clock,** which is the most common setting for single-axis scans, the estimated probe speed that you enter is used to determine the position of each data point.

Acquisition density and resolution are automatically calculated based on the entered parameters.

5. In the **Strip chart** section, select the data that you want to view, and how it should be displayed.

Figure 3-38 The **Strip chart** section



   a. Select whether the data is time-based (in seconds), distance-based (in mm), or sample-based (samples/pixel).
      Distance-based strip charts are available when using encoders or non-standard clocks.
   b. Enter the vertical scale that you want to use. The unit indicated is linked to the type of data selected in the previous substep (time, distance, or sample).
   c. In the **Direction** subsection, select whether new data is added to the bottom of the strip chart. For more information on data direction, see page 45

6. When you are done setting up your scan parameters, click **OK**. Scan parameters are saved, the **Scan Parameters** window closes, and you return to Magnifi. Click **Cancel** to discard all the modifications that you made.

For Tube/Bore Scans with Rotating Probes

To set up scan parameters for tube/bore scans with rotating probes (*and for IRIS inspections*):

1. Select **Scan Parameters** from the **Settings** menu. The **Scan Parameters** window appears.

Figure 3-39 The **Scan Parameters** window for tube/bore scans with rotating probes

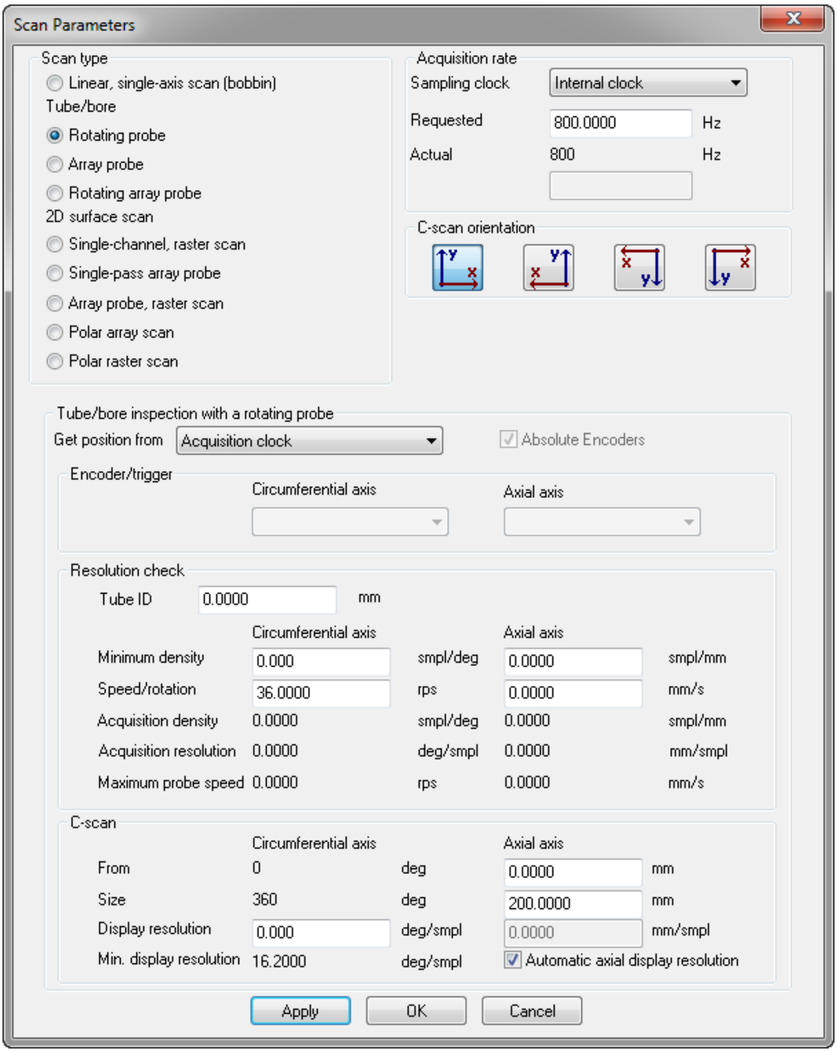
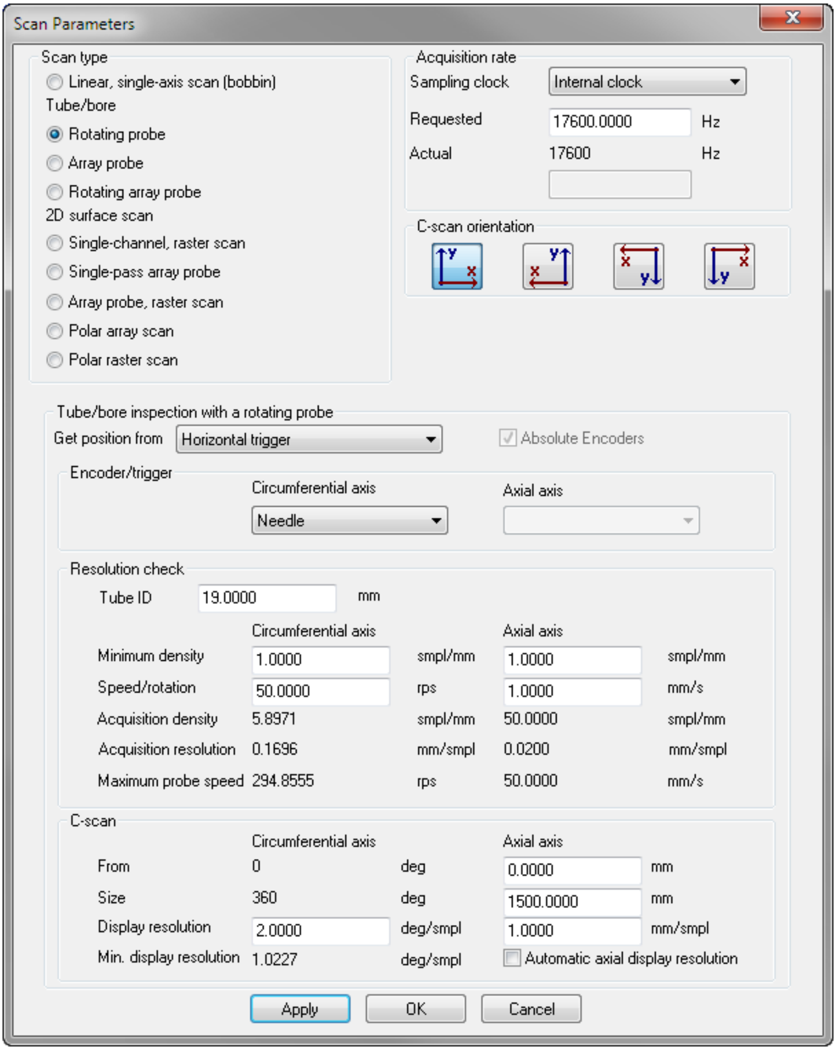
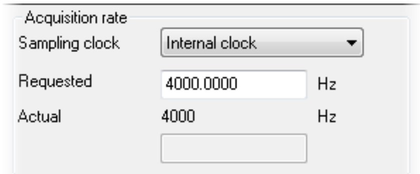


Figure 3-40 The **Scan Parameters** window for typical IRIS scans



2. In the **Scan type** section, select **Rotating probe** under **Tube/bore**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder). For more information on defining encoders, see page 82.

Figure 3-41 The **Acquisition rate** section: **Internal clock** selected



a From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.

b If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.

In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.

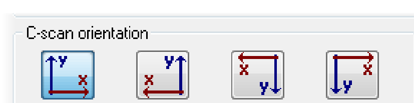
c If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.

d If you chose a preset encoder, you have to enter the requested acquisition rate in the **Requested** field, and the maximum acquisition rate in the **Max. acquisition rate** field.

Note *Depending on how the encoder is defined, acquisition rate measurement units might differ from the ones selected for the application in the* ***General*** *tab of the* ***Preferences*** *dialog box (see “Setting Measurement Units” on page 73).*

4. In the **C-scan orientation** section, select the button representing the orientation that you want to use for C-scans’ X and Y axes. This only affects how data is displayed; is has no effect on the scan itself.

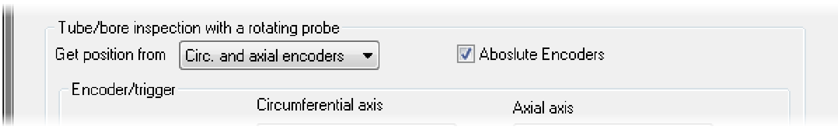
Figure 3-42 The **C-scan orientation** section



Important **In a tube/bore configuration, the X axis represents the circumference, while the Y axis represents the length of the tube.**

5. In the **Tube/bore inspection with a rotating probe** section, configure your basic scan parameters.

Figure 3-43 The **Tube/bore inspection with a rotating probe** section



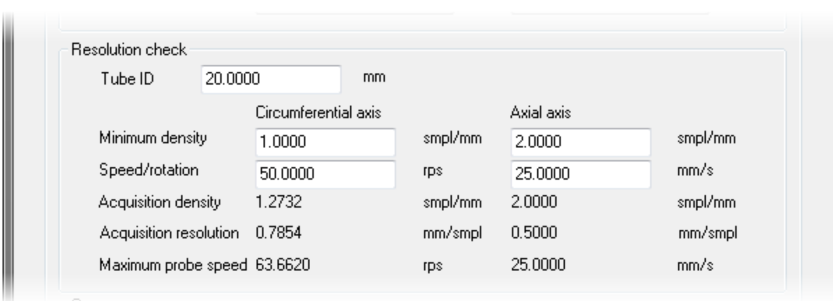
- **a** In the **Get position from** drop-down list, select what device will provide probe positions. The list of available devices depends on the selection made previously in the **Sampling clock** drop-down list.

  If you selected **Internal clock** or **External clock** from the **Sampling clock** drop-down list, the following choices are available: **Circ. and axial encoders, Circumferential encoder, Vertical Trigger And Axial Encoder, Horizontal trigger, Vertical trigger, Index channel, Horizontal Trigger And Axial Encoder, Acquisition clock.**

  If you selected a preset encoder in the **Sampling clock** list, only the following are available: **Circ. and axial encoders**, **Circumferential encoder**, **Acquisition clock**, **Horizontal Trigger and Axial Encoder**, and **Vertical Trigger and Axial Encoder**.
- **b** If you selected encoders from the **Get position from** drop-down list, check the **Absolute Encoders** box, if necessary. When this option is enabled, the position is always positive.

6. If you selected an encoder in step 5a, select the appropriate encoder(s) for the circumferential and/or axial axes in the **Encoder/trigger** subsection.
7. In the **Resolution check** subsection, define the resolution parameters that will be used during acquisitions.

Figure 3-44 The **Resolution check** subsection

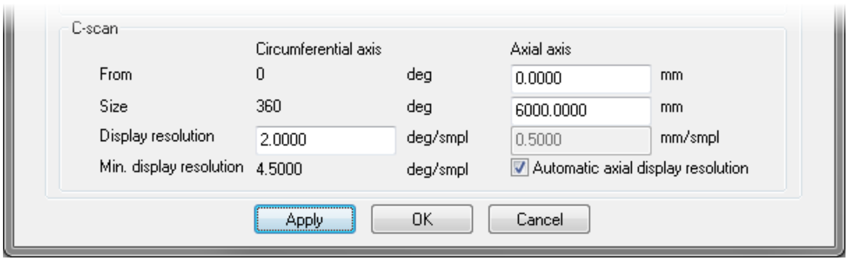


- **a** Enter the tube internal diameter in the **Tube ID** field.
- **b** In the **Minimum density** fields (under the **Circumferential axis** and **Axial axis** columns), enter the number of samples that you want to take per unit of distance.
- **c** In the **Speed/rotation** fields (under the **Circumferential axis** and **Axial axis** columns), enter the speed at which the probe will travel along both axes.

Once all this data is entered, Magnifi calculates acquisition density and resolution, as well as the maximum probe speed.

8. In the **C-scan** subsection, define the range of values used to display C-scans in a data pane.

Figure 3-45 The **C-scan** subsection



- **a** In the **Display resolution** field under the **Circumferential axis** column, enter the angle variation required between samples.
- **b** In the **From** field under the **Axial axis** column, enter the starting point for the C-scan display, based on the size of the piece under test.
- **c** In the **Size** field under the **Axial axis** column, enter the end point for the C-scan display, based on the size of the piece under test. Normally, the value entered in this field is the total size of the piece under test.
- **d** By default the display resolution on the axial axis is calculated automatically. If you want to enter a specific display resolution for this axis, uncheck the **Automatic axial display resolution** box. This activates the **Display resolution** field under the **Axial axis** column. Enter the required resolution in that field.

9. Click **OK** when you are done setting up your scan parameters. Scan parameters are saved, the **Scan Parameters** window closes, and you return to the Magnifi application. Click **Cancel** to discard all the modifications that you made.

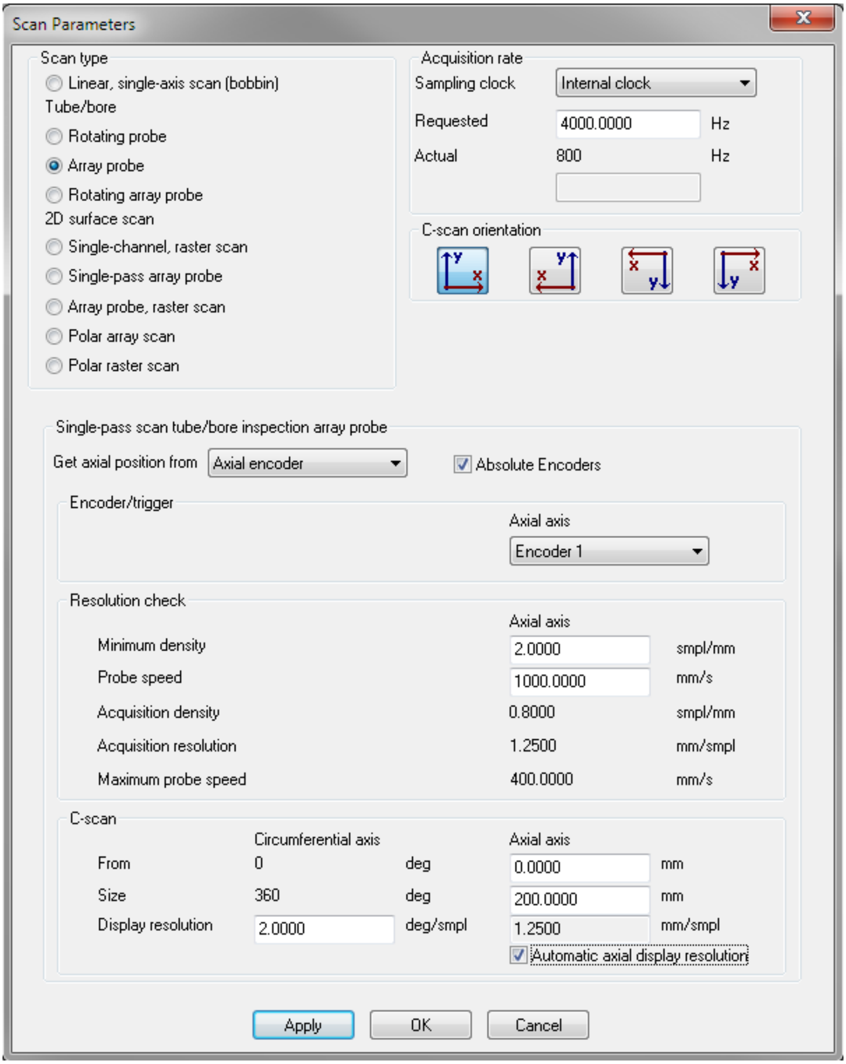
Note *For a typical example on how to set for an IRIS inspection, look carefully at Figure 3-40.*

For Tube/Bore Scans with Array Probes

To set up scan parameters for tube/bore scans with array (DefHi™, etc.) probes:

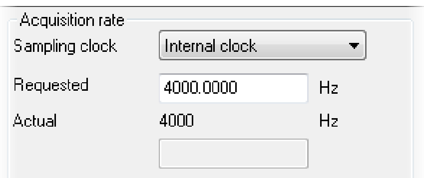
1. Select **Scan Parameters** from the **Settings** menu. The **Scan Parameters** window appears.

Figure 3-46 The **Scan Parameters** window for tube/bore scans with array probes



2. In the **Scan type** section, select **Array probe** under **Tube/bore**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder). For more information on defining encoders, see page 82.

Figure 3-47 The **Acquisition rate** section: **Internal clock** selected



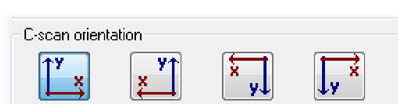
a. From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.
b. If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.

   In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.
c. If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.
d. If you chose a preset encoder, you have to enter the requested acquisition rate in the **Requested** field, and the maximum acquisition rate in the **Max. acquisition rate** field.

Note *Depending on how the encoder is defined, acquisition rate measurement units might differ from the ones selected for the application in the **General** tab of the **Preferences** dialog box (see "Setting Measurement Units" on page 73).*

4. In the **C-scan orientation** section, select the button representing the orientation that you want to use for C-scans' X and Y axes. This only affects how data is displayed; is has no effect on the scan itself.

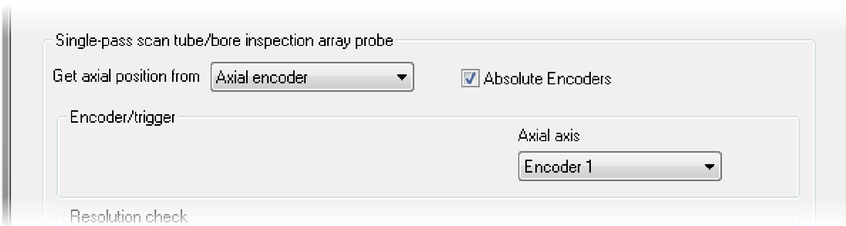
Figure 3-48 The **C-scan orientation** section



Important In a tube/bore configuration, the X axis represents the circumference, while the Y axis represents the length of the tube.

5. In the **Single-pass scan tube/bore inspection array probe** section, configure your basic scan parameters.

Figure 3-49 The **Single-pass scan tube/bore inspection array probe** section



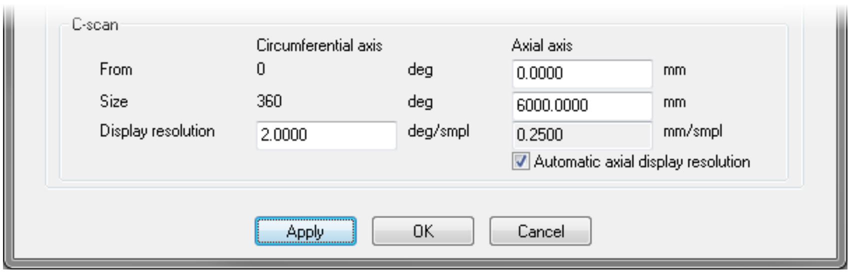
   a In the **Get axial position from** drop-down list, select what device will provide probe positions (**Acquisition clock, Non-standard clock, Axial encoder**).
   b If you selected **Axial encoder** from the **Get axial position from** drop-down list, check the **Absolute Encoders** box, if necessary. This option allows you to use negative and positive position values.
6. If you selected an encoder in step 5a, select the appropriate encoder for the axial axis in the **Encoder/trigger** subsection.
7. In the **Resolution check** subsection, define the resolution parameters that will be used during acquisitions.

Figure 3-50 The **Resolution check** subsection

   a In the **Minimum density** field, enter the number of samples that you want to take per unit of distance.
   b In the **Probe speed** field, enter the speed (in **in./s** or **mm/s**) at which the probe will travel along both axes.
   Once all this data is entered, Magnifi calculates acquisition density and resolution, as well as the maximum probe speed.

8. In the **C-scan** subsection, define the range of values used to display C-scans in a data pane.

Figure 3-51 The **C-scan** subsection



a In the **Display resolution** field under the **Circumferential axis** column, enter the angle variation required between samples.
b In the **From** field under the **Axial axis** column, enter the starting point for the C-scan display, based on the size of the piece under test.
c In the **Size** field under the **Axial axis** column, enter the end point for the C-scan display, based on the size of the piece under test. Normally, the value entered in this field is the total size of the piece under test.
d By default the display resolution on the axial axis is calculated automatically. If the value indicated is not the one that you expected, uncheck the **Automatic axial display resolution** box. This activates the **Display resolution** field under the **Axial axis** column. Enter the required resolution in that field.

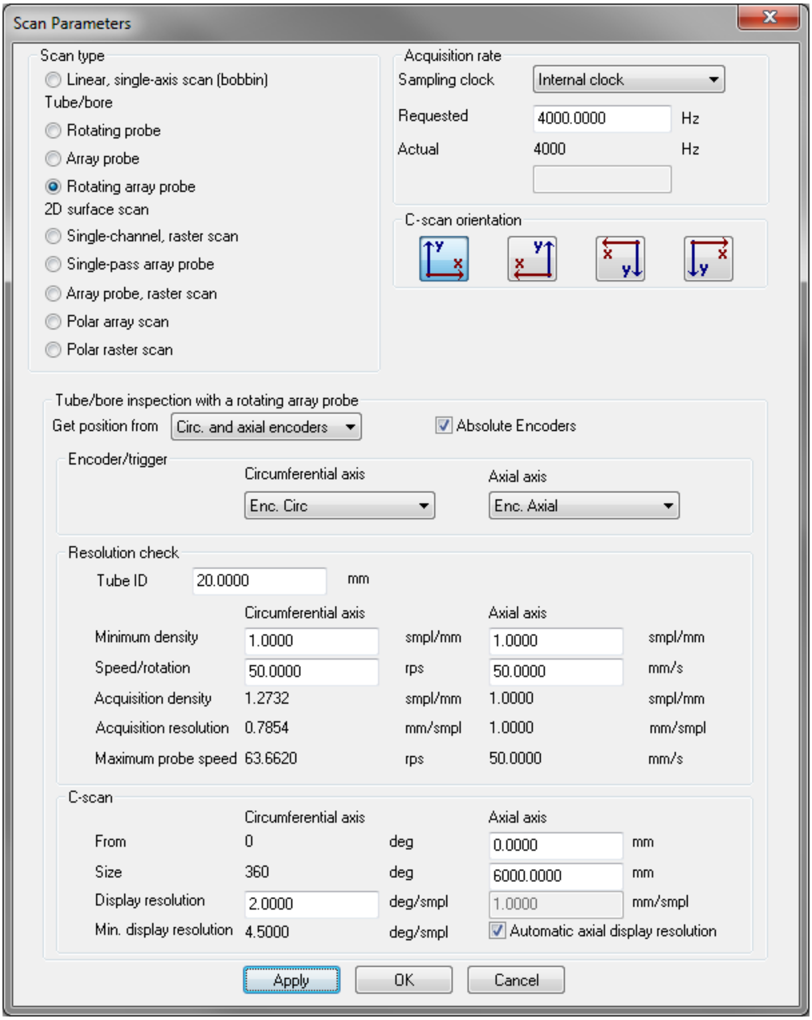
9. Click **OK** when you are done setting up your scan parameters. Scan parameters are saved, the **Scan Parameters** window closes, and you return to the Magnifi application. Click **Cancel** to discard all the modifications that you made.

For Tube/Bore Scans with Rotating Array Probes

To set up scan parameters for tube/bore scans with rotating array probes:

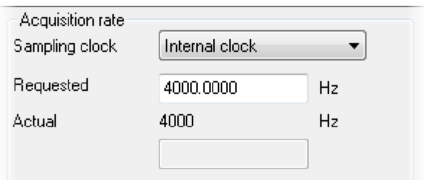
1. Select **Scan Parameters** from the **Settings** menu. The **Scan Parameters** window appears.

Figure 3-52 The **Scan Parameters** window for tube/bore scans with rotating array probes



2. In the **Scan type** section, select **Rotating array probe** under **Tube/bore**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder). For more information on defining encoders, see page 82.

Figure 3-53 The **Acquisition rate** section: **Internal clock** selected



a From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.

b If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.

In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.

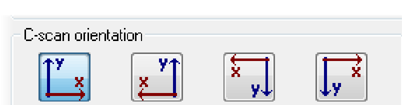
c If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.

d If you chose a preset encoder, you have to enter the requested acquisition rate in the **Requested** field, and the maximum acquisition rate in the **Max. acquisition rate** field.

Note *Depending on how the encoder is defined, acquisition rate measurement units might differ from the ones you selected for the application in the **General** tab of the **Preferences** dialog box (see "Setting Measurement Units" on page 73).*

4. In the **C-scan orientation** section, select the button representing the orientation that you want to use for C-scans' X and Y axes. This only affects how data is displayed; is has no effect on the scan itself.

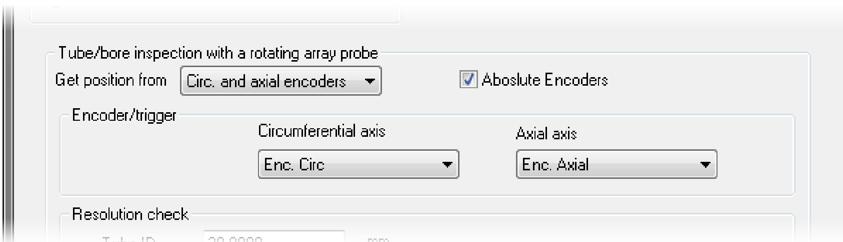
Figure 3-54 The **C-scan orientation** section



Important In a tube/bore configuration, the X axis represents the circumference, while the Y axis represents the length of the tube.

5. In the **Tube/bore inspection with a rotating array probe** section, configure your basic scan parameters.

Figure 3-55 The **Tube/bore inspection with a rotating array probe** section



a In the **Get position from** drop-down list, select what device will provide probe positions. The list of available devices depends on the selection made previously in the **Sampling clock** drop-down list.

If you selected **Internal clock** or **External clock** from the **Sampling clock** list, the following choices are available: **Circ. and axial encoders, Circumferential encoder, Vertical Trigger And Axial Encoder, Horizontal Trigger And Axial Encoder, Acquisition clock.**

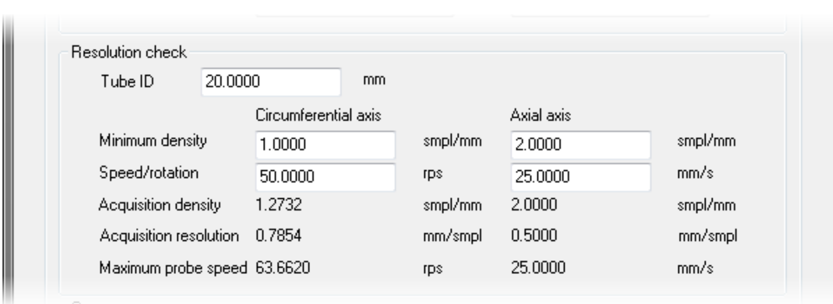
If you selected a preset encoder in the **Sampling clock** list, only the first three choices are available **(Circ. and axial encoders, Circumferential encoder, Acquisition clock)**.

b If you selected encoders from the **Get position from** drop-down list, check the **Absolute Encoders** box, if necessary. This option allows you to use negative and positive position values.

6. If you selected an encoder in step 5a, select the appropriate encoder(s) for the circumferential and/or axial axes in the **Encoder/trigger** subsection.

7. In the **Resolution check** subsection, define the resolution parameters that will be used during acquisitions.

Figure 3-56 The **Resolution check** subsection



a Enter the tube internal diameter in the **Tube ID** field.

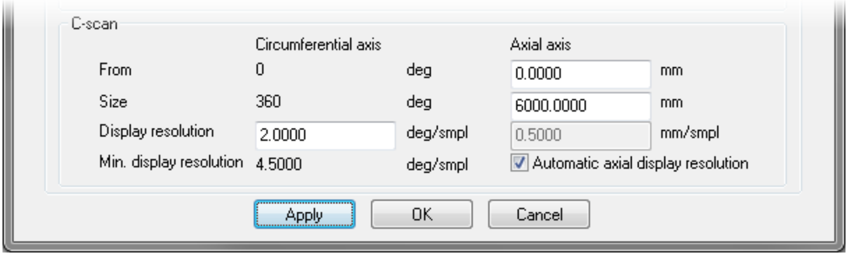
b In the **Minimum density** fields (under the **Circumferential axis** and **Axial axis** columns), enter the number of samples that you want to take per unit of distance.

c In the **Speed/rotation** fields (under the **Circumferential axis** and **Axial axis** columns), enter the speed (in **in./s** or **mm/s**) at which the probe will travel along both axes.

Once all this data is entered, Magnifi calculates acquisition density and resolution, as well as the maximum probe speed.

8. In the **C-scan** subsection, define the range of values used to display C-scans in a data pane.

Figure 3-57 The **C-scan** subsection



a In the **Display resolution** field under the **Circumferential axis** column, enter the angle variation required between samples.

b In the **From** field under the **Axial axis** column, enter the starting point for the C-scan display, based on the size of the piece under test.

c In the **Size** field under the **Axial axis** column, enter the end point for the C-scan display, based on the size of the piece under test. Normally, the value entered in this field is the total size of the piece under test.

d By default the display resolution on the axial axis is calculated automatically. If the value indicated is not the one that you expected, uncheck the **Automatic axial display resolution** box. This activates the **Display resolution** field under the **Axial axis** column. Enter the required resolution in that field.

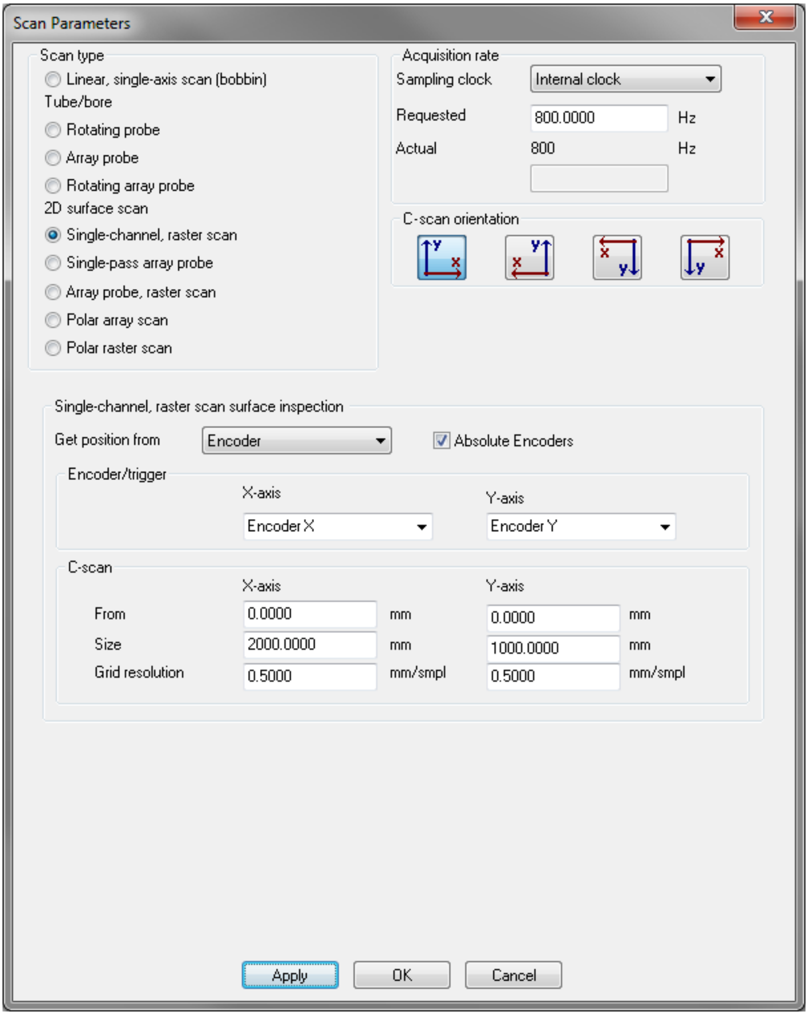
9. Click **OK** when you are done setting up your scan parameters. Scan parameters are saved, the **Scan Parameters** window closes, and you return to the Magnifi application. Click **Cancel** to discard all the modifications that you made.

For 2D Surface Raster Scans (Single-Channel)

To set up scan parameters for 2D single-channel surface raster scans:

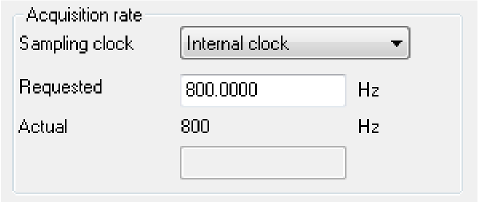
1. Select **Scan Parameters** from the **Settings** menu. The **Scan Parameters** window appears.

Figure 3-58 The **Scan Parameters** window for 2D single-channel surface raster scans



2. In the **Scan type** section, select **Single-channel, raster scan** under **2D surface scan**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder). For more information on defining encoders, see page 82.

Figure 3-59 The **Acquisition rate** section: **Internal clock** selected



a From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.

b If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.
In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.

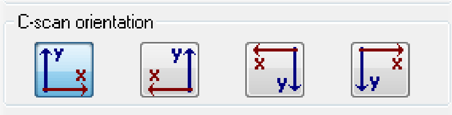
c If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.

d If you chose a preset encoder, you have to enter the requested acquisition rate in the **Requested** field, and the maximum acquisition rate in the **Max. acquisition rate** field.

Note *Depending on how the encoder is defined, acquisition rate measurement units might differ from the ones you selected for the application in the* ***General*** *tab of the* ***Preferences*** *dialog box (see "Setting Measurement Units" on page 73).*

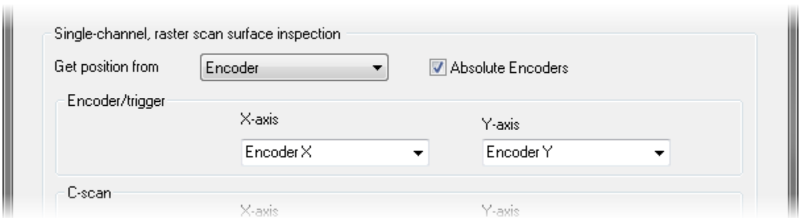
4. In the **C-scan orientation** section, select the button representing the orientation that you want to use for C-scans' X and Y axes. This only affects how data is displayed; it has no effect on the actual scan.

Figure 3-60 The **C-scan orientation** section



5. In the **Single-channel, raster scan surface inspection** section configure your basic scan parameters.

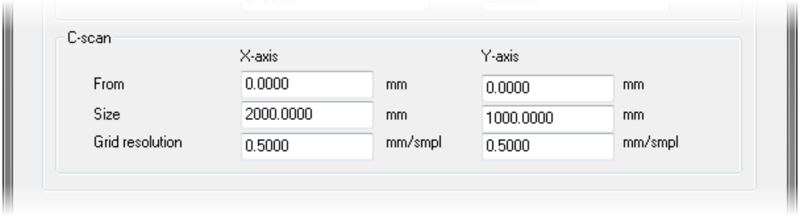
Figure 3-61 The **Single-channel, raster scan surface inspection** section



   a In the **Get position from** drop-down list, **Encoders** is selected. You have no other choice. Check the **Absolute Encoders** box, if necessary. This option allows you to use negative and positive position values.

6. In the **Encoder/trigger** subsection, select the appropriate encoder for the X and Y axes.
7. In the **C-scan** subsection, define the range of values used to display C-scans in a data pane.

Figure 3-62 The **C-scan** subsection



   a In the **From** fields, enter the starting point for the C-scan display on the X and Y axes, based on the size of the piece under test.
   b In the **Size** fields, enter the end point for the C-scan display on the X and Y axes, based on the size of the piece under test. Normally, the value entered in this field is the total size of the piece under test.
   c In the **Grid resolution** columns, enter the length and width of each sample on the grid.

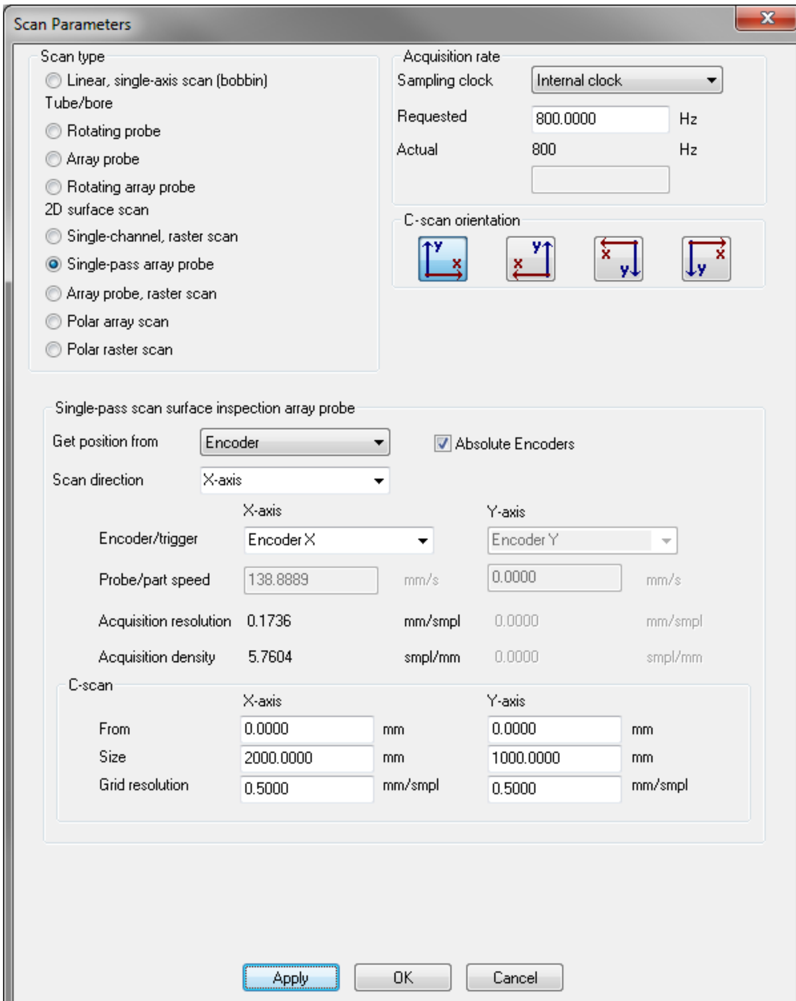
8. Click **OK** when you are done setting up your scan parameters. Scan parameters are saved, the **Scan Parameters** window closes, and you return to the Magnifi application. Click **Cancel** to discard all the modifications that you made.

For 2D Surface Scans with Single-Pass Array Probes

To set up scan parameters for 2D surface scans with single-pass array probes:

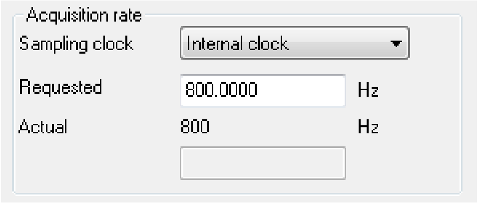
1. Select **Scan Parameters** from the **Settings** menu. The **Scan Parameters** window appears.

Figure 3-63 The **Scan Parameters** window for 2D surface scans with single-pass array probes



2. In the **Scan type** section, select **Single-pass array probe** under **2D surface scan**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder). For more information on defining encoders, see page 82.

Figure 3-64 The **Acquisition rate** section: **Internal clock** selected



a From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.

b If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.
In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.

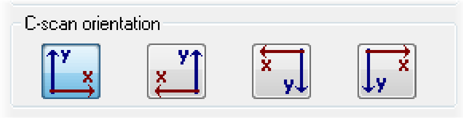
c If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.

d If you chose a preset encoder, you have to enter the requested acquisition rate in the **Requested** field, and the maximum acquisition rate in the **Max. acquisition rate** field.

Note *Depending on how the encoder is defined, acquisition rate measurement units might differ from the ones you selected for the application in the* ***General*** *tab of the* ***Preferences*** *dialog box (see "Setting Measurement Units" on page 73).*

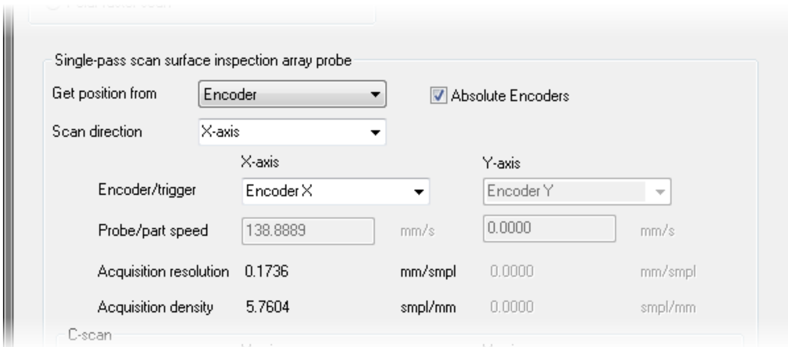
4. In the **C-scan orientation** section, select the button representing the orientation that you want to use for C-scans' X and Y axes. This only affects how data is displayed; is has no effect on the scan itself.

Figure 3-65 The **C-scan orientation** section



5. In the **Single-pass scan surface inspection array probe** section, configure your basic scan parameters.

Figure 3-66 The **Single-pass scan surface inspection array probe** section



a In the **Get position from** drop-down list, select what device will provide probe positions. The list of available devices depends on the selection made previously in the **Sampling clock** drop-down list.

If you selected **Internal clock** or **External clock** from the **Sampling clock** drop-down list, the following choices are available: **Encoder, Non-standard clock, Acquisition clock.**

If you selected a preset encoder in the **Sampling clock** drop-down list, only **Encoder** is available.

b If you selected encoders from the **Get position from** drop-down list, check the **Absolute Encoders** box, if necessary. This option allows you to use negative and positive position values.

c In the **Scan direction** drop-down list, select the axis along which the acquisitions will be performed. Depending on this selection, the relevant fields in the columns underneath will activate.

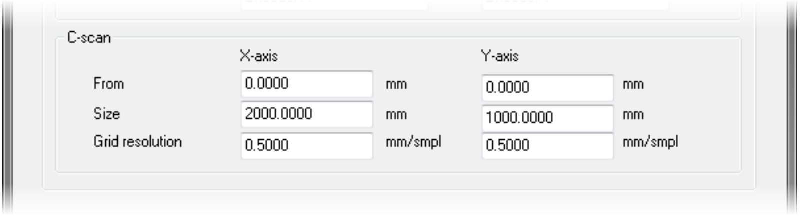
d If you selected **Encoder** from the **Get position from** drop-down list, select the appropriate encoder from the **Encoder/trigger** drop-down list.

e If you selected *anything but* **Encoder** from the **Get position from** drop-down list, enter the probe speed in the **Probe/part speed** field.

Acquisition resolution and density are automatically calculated based on the entered values.

6. In the **C-scan** subsection, define the range of values used to display C-scans in a data pane.

Figure 3-67 The **C-scan** subsection



a In the **From** fields, enter the starting point for the C-scan display on the X and Y axes, based on the size of the piece under test.

b In the **Size** fields, enter the end point for the C-scan display on the X and Y axes, based on the size of the piece under test. Normally, the value entered in this field is the total size of the piece under test.

c In the **Grid resolution** columns, enter the length and width of each sample on the grid.

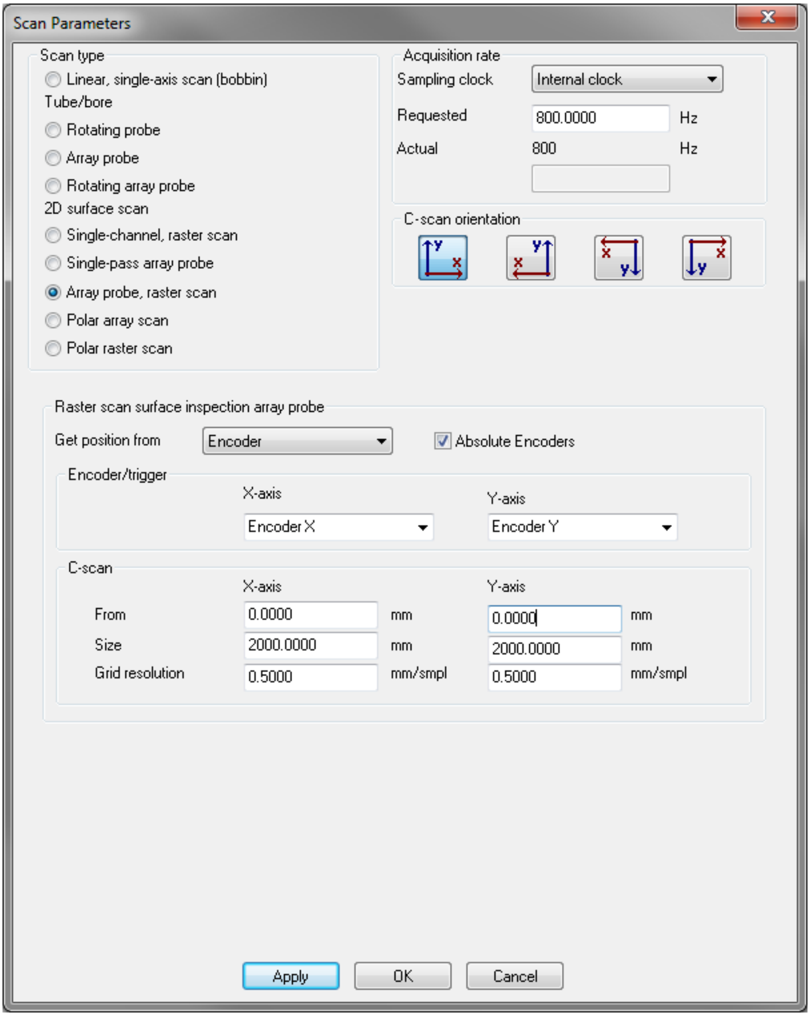
7. Click **OK** when you are done setting up your scan parameters. Scan parameters are saved, the **Scan Parameters** window closes, and you return to the Magnifi application. Click **Cancel** to discard all the modifications that you made.

For 2D Surface with Raster Scan Array Probes

To set up scan parameters for 2D surface raster scans with array probes:

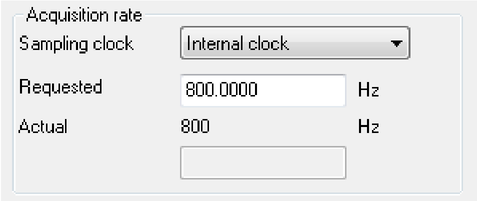
1. Select **Scan Parameters** from the **Settings** menu. The **Scan Parameters** window appears.

Figure 3-68 The **Scan Parameters** window for 2D surface with raster scan array probes



2. In the **Scan type** section, select **Array probe, raster scan** under **2D surface scan**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder). For more information on defining encoders, see page 82.

Figure 3-69 The **Acquisition rate** section: **Internal clock** selected



a From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.

b If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.
In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.

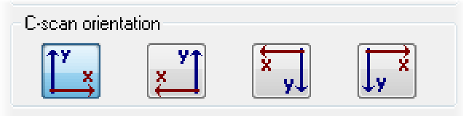
c If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.

d If you chose a preset encoder, you have to enter the requested acquisition rate in the **Requested** field, and the maximum acquisition rate in the **Max. acquisition rate** field.

Note *Depending on how the encoder is defined, acquisition rate measurement units might differ from the ones you selected for the application in the* ***General*** *tab of the* ***Preferences*** *dialog box (see "Setting Measurement Units" on page 73).*

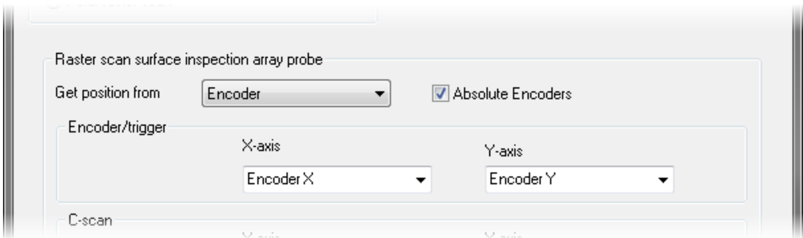
4. In the **C-scan orientation** section, select the button representing the orientation that you want to use for C-scans' X and Y axes. This only affects how data is displayed; is has no effect on the scan itself.

Figure 3-70 The **C-scan orientation** section



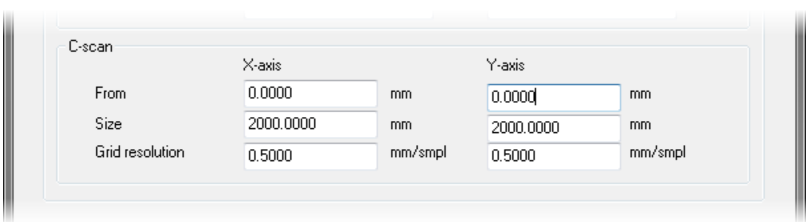
5. In the **Raster scan surface inspection array probe** section, configure your basic scan parameters. In the **Get position from** drop-down list, **Encoders** is selected. You have no other choice. Check the **Absolute Encoders** box, if necessary. This option allows you to use negative and positive position values.

Figure 3-71 The **Raster scan surface inspection array probe** section



6. In the **Encoder/trigger** subsection, select the appropriate encoder for the X and Y axes.
7. In the **C-scan** subsection, define the range of values used to display C-scans in a data pane.

Figure 3-72 The **C-scan** subsection



   a In the **From** fields, enter the starting point for the C-scan display on the X and Y axes, based on the size of the piece under test.
   b In the **Size** fields, enter the end point for the C-scan display on the X and Y axes, based on the size of the piece under test. Normally, the value entered in this field is the total size of the piece under test.
   c In the **Grid resolution** columns, enter the length and width of each sample on the grid.

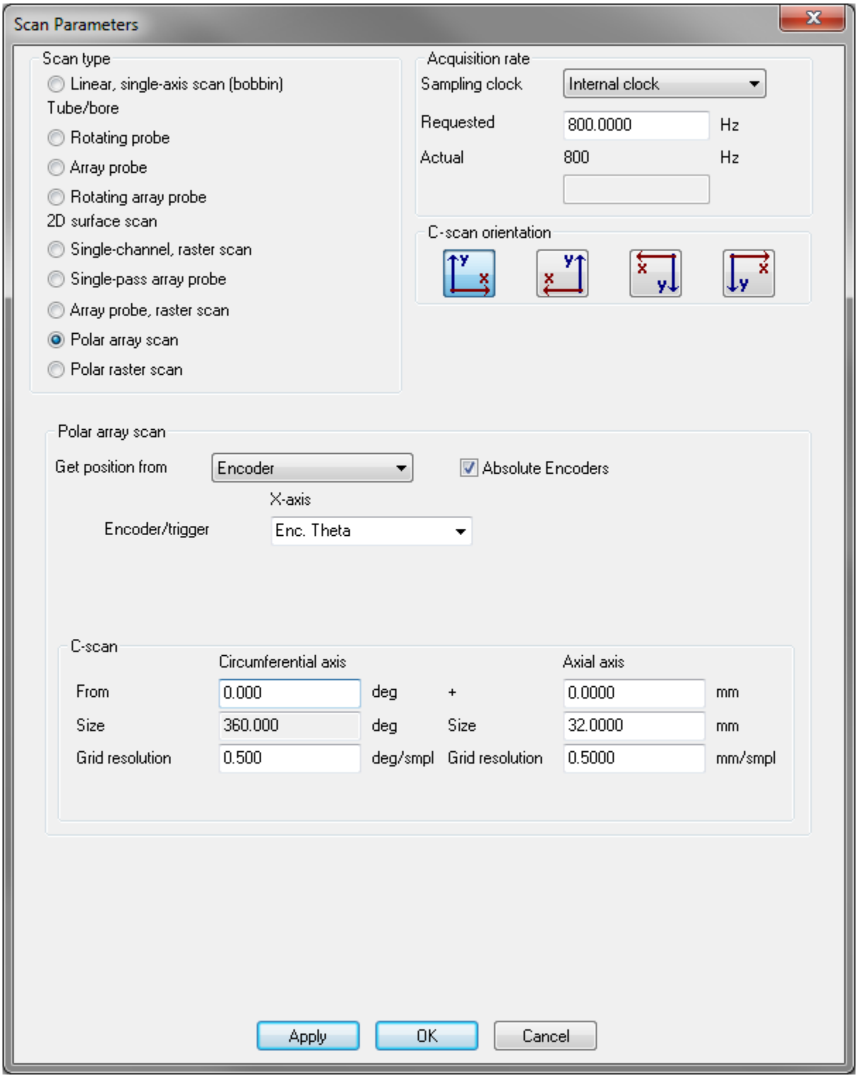
8. Click **OK** when you are done setting up your scan parameters. Scan parameters are saved, the **Scan Parameters** window closes, and you return to the Magnifi application. Click **Cancel** to discard all the modifications that you made.

For 2D Surface Polar Array Scans

To set up scan parameters for 2D surface array polar scans:

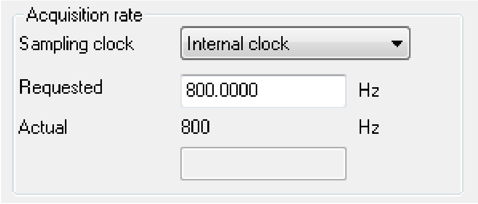
1. Select **Scan Parameters** from the **Settings** menu. The **Scan Parameters** window appears.

Figure 3-73 The **Scan Parameters** window for 2D surface array polar scans



2. In the **Scan type** section, select **Polar array scan** under **2D surface scan**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder). For more information on defining encoders, see page 82.

Figure 3-74 The **Acquisition rate** section: **Internal clock** selected

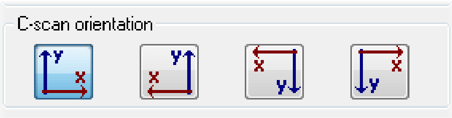


   a From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.
   b If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.
   In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.
   c If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.
   d If you chose a preset encoder, you have to enter the requested acquisition rate in the **Requested** field, and the maximum acquisition rate in the **Max. acquisition rate** field.

Note *Depending on how the encoder is defined, acquisition rate measurement units might differ from the ones selected for the application in the* ***General*** *tab of the* ***Preferences*** *dialog box (see "Setting Measurement Units" on page 73).*

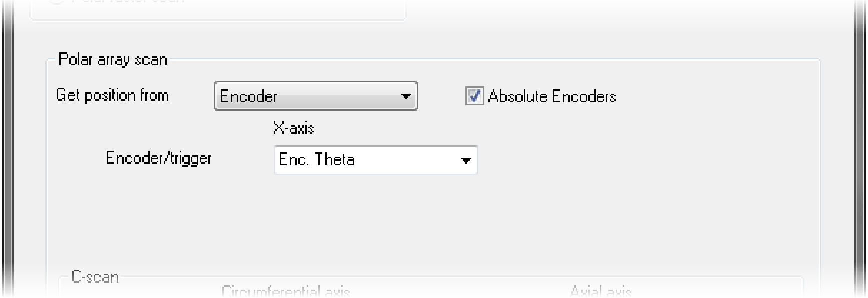
4. In the **C-scan orientation** section, select the button representing the orientation that you want to use for C-scans' X and Y axes. This only affects how data is displayed; is has no effect on the scan itself.

Figure 3-75 The **C-scan orientation** section



5. In the **Polar array scan** section, configure your basic scan parameters.

Figure 3-76 The **Polar array scan** section



a In the **Get position from** drop-down list, select what device will provide probe positions. The list of available devices depends on the selection made previously in the **Sampling clock** drop-down list.

If you selected **Internal clock** or **External clock** from the **Sampling clock** drop-down list, the following choices are available: **Encoder, Non-standard clock, Acquisition clock.**

If you selected a preset encoder in the **Sampling clock** drop-down list, only **Encoder** is available.

b If you selected **Acquisition clock** from the **Get position from** drop-down list, go to step 6.

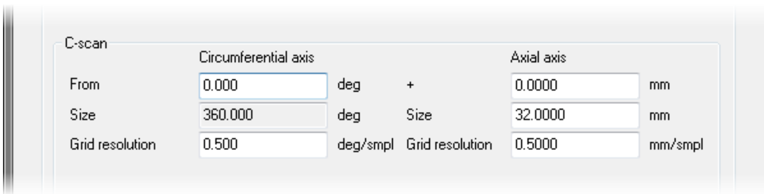
If you selected **Non-standard clock** from the **Get position from** drop-down list, enter the speed at which the probe/part will move along the X axis in the **Probe/part speed** field.

If you selected **Encoder** from the **Get position from** drop-down list, select the encoder that corresponds to the angular path in the **Encoder/trigger** drop-down list. Check the **Absolute Encoders** box, if necessary. This option allows you to use negative and positive position values.

Note *If the encoder providing the angular position is set in millimeters or inches, you will have to specify the encoder path diameter in the* ***Encoder path diameter*** *field (this information is used to convert the length in degrees). If the encoder is already set in degrees, you will have nothing to specify.*

6. In the **C-scan** subsection, define the range of values used to display C-scans in a data pane.

Figure 3-77 The **C-scan** subsection



Under the **Circumferential axis** column, enter the scan starting point, in degrees, in the **From** field, and the resolution that you want to use for your grid (the number of degrees covered to constitute a sample) in the **Grid resolution** field.

Under the **Axial axis** column, enter the starting radius of the inspected area in the "**+**" field, the external radius of the polar scan in the **Size** field, and the resolution that you want to use for your grid (the distance covered along the X axis to constitute a sample) in the **Grid resolution** field.

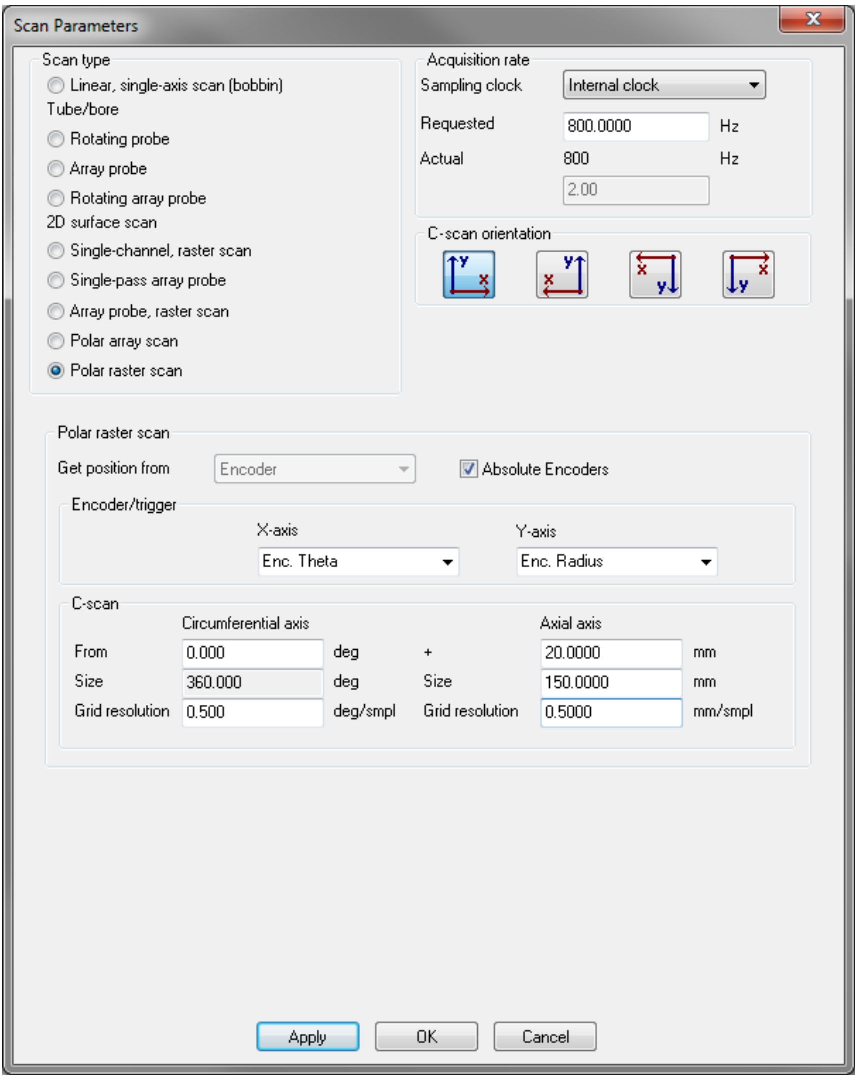
7. Click **OK** when you are done setting up your scan parameters. Scan parameters are saved, the **Scan Parameters** window closes, and you return to the Magnifi application. Click **Cancel** to discard all the modifications that you made.

For 2D Surface Polar Raster Scans

To set up scan parameters for 2D surface polar raster scans:

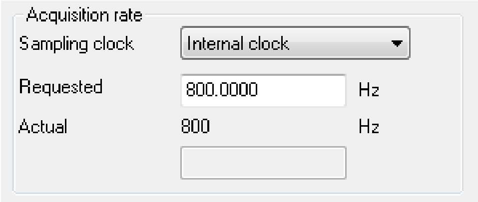
1. Select **Scan Parameters** from the **Settings** menu. The **Scan Parameters** window appears.

Figure 3-78 The **Scan Parameters** window for 2D surface polar raster scans



2. In the **Scan type** section, select **Polar raster scan** under **2D surface scan**. When you make your selection, the bottom part of the **Scan Parameters** window changes to show parameters relevant to this type of scan.
3. In the **Acquisition rate** section, select how the acquisition rate will be controlled (internal clock, external clock, or predefined encoder). For more information on defining encoders, see page 82.

Figure 3-79 The **Acquisition rate** section: **Internal clock** selected



a From the **Sampling clock** drop-down list, select whether the acquisition rate is controlled by an internal clock, an external clock, or a preset encoder.

b If you chose **Internal clock,** enter the acquisition rate (in Hz) that you want to use in the **Requested** field.

In certain circumstances, because of limitations of the *connected* instrument, the acquisition rate that you request might be unreachable. The software will then automatically calculate the closest possible rate, and indicate the calculated rate in the **Actual** field.

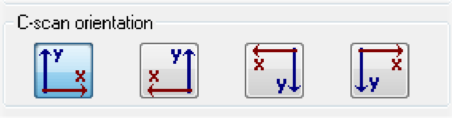
c If you chose **External clock,** the **Requested** field is inactive, but the **Max. acquisition rate** field appears, where you instruct the external clock about the maximum acquisition rate (in Hz) that you want to use.

d If you chose a preset encoder, you have to enter the requested acquisition rate in the **Requested** field, and the maximum acquisition rate in the **Max. acquisition rate** field.

Note *Depending on how the encoder is defined, acquisition rate measurement units might differ from the ones selected for the application in the **General** tab of the **Preferences** dialog box (see "Setting Measurement Units" on page 73).*

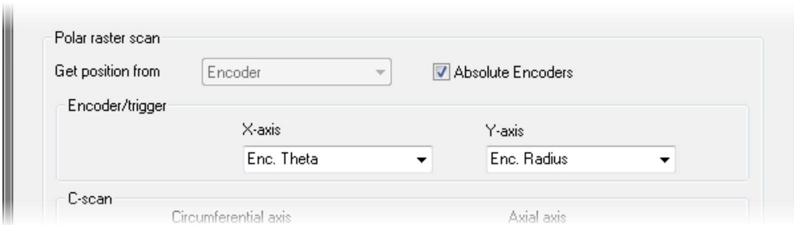
4. In the **C-scan orientation** section, select the button representing the orientation that you want to use for C-scans' X and Y axes. This only affects how data is displayed; is has no effect on the scan itself.

Figure 3-80 The **C-scan orientation** section



5. In the **Polar raster scan** section, configure basic scan parameters. In the **Get position from** drop-down list, **Encoders** is selected. You have no other choice. Check the **Absolute Encoders** box, if necessary.

Figure 3-81 The **Polar raster scan** section



6. In the **Encoder/trigger** subsection, select the appropriate encoder for the X and Y axes.
7. In the **C-scan** subsection, define the range of values used to display C-scans in a data pane.

Figure 3-82 The **C-scan** subsection

a Under the **Circumferential axis** column, enter the scan starting point, in degrees, in the **From** field.
b Enter the resolution that you want to use for your grid (the number of degrees covered to constitute a sample) in the **Grid resolution** field.
c Under the **Axial axis** column, enter the starting radius of the inspected area in the "**+**" field.
d Enter the external radius of the polar scan in the **Size** field.
e Enter the resolution that you want to use for your grid (the distance covered along the X axis to constitute a sample) in the **Grid resolution** field.

8. Click **OK** when you are done setting up your scan parameters. Scan parameters are saved, the **Scan Parameters** window closes, and you return to the Magnifi application. Click **Cancel** to discard all the modifications that you made.

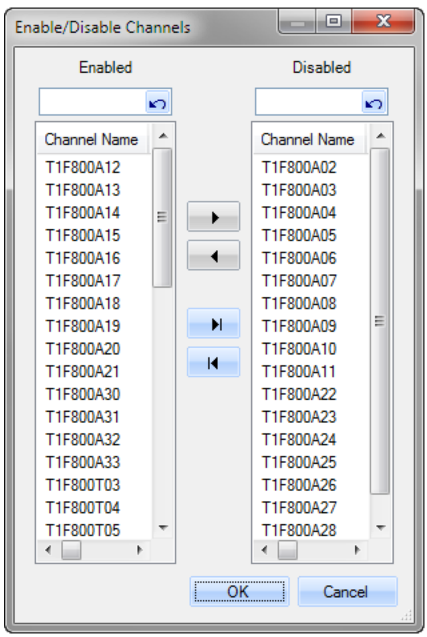
Enabling/Disabling Channels

Before performing acquisitions, you might want to select specific channels. For this, you need to know how to enable and disable channels.

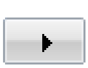
To disable selected channels:

1. From the **Settings** menu, select **Acquisition Setup>Enable/Disable Channels**, or click the **Enable/disable channels** button in the Acquisition Setup toolbar. The **Enable/Disable Channels** window appears.

Figure 3-83 The **Enable/Disable Channels** window



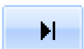
2. In the **Enabled** list, select the channels to disable:
   - To select one channel, click the channel.
   - To select contiguous channels in the list, click one channel, press the SHIFT key, and click the last channel that you want to select. All channels between the first and last selected become selected as well.
   - To select non-contiguous channels in the list, press the CTRL key while clicking on all the channels that you want to select.



3. Click the **Disable the selected channel(s)** button to send the selected channels in the **Disabled** column.
4. Click **OK** when you are done.

To disable all channels:

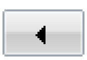
1. From the **Settings** menu, select **Acquisition Setup>Enable/Disable Channels,** or click the **Enable/disable channels** button in the Acquisition Setup toolbar. The **Enable/Disable Channels** window appears.



2. Click the **Disable all filtered channel(s)** button to send all channels in the **Disabled** column.
3. Click **OK** when you are done.

To enable selected channels:

1. From the **Settings** menu, select **Acquisition Setup>Enable/Disable Channels,** or click the **Enable/disable channels** button in the Acquisition Setup toolbar. The **Enable/Disable Channels** window appears.
2. In the **Disabled** list (see Figure 3-83), select the channels to enable:
   - To select one channel, click the channel.
   - To select contiguous channels in the list, click one channel, press the SHIFT key, and click the last channel that you want to select. All channels between the first and last selected become selected as well.
   - To select non-contiguous channels in the list, press the CTRL key while clicking on all the channels that you want to select.



3. Click the **Enable selected channel(s)** button to send the selected channels in the **Enabled** column.
4. Click **OK** when you are done.

To enable all channels:

1. From the **Settings** menu, select **Acquisition Setup>Enable/Disable Channels,** or click the **Enable/disable channels** button in the Acquisition Setup toolbar. The **Enable/Disable Channels** window appears.

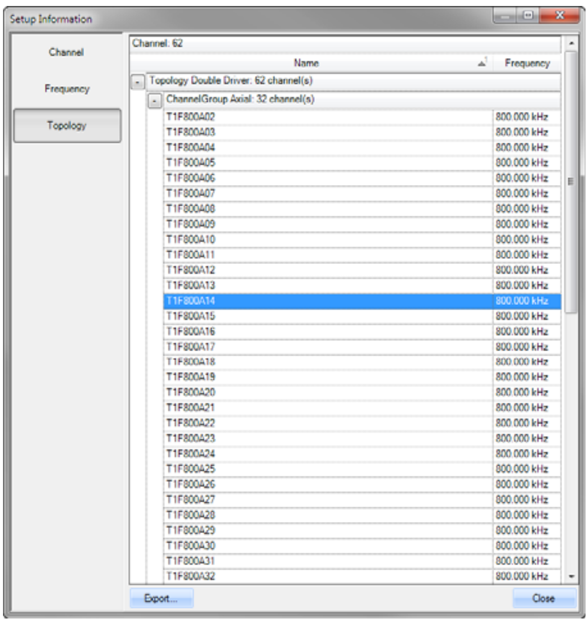
2. Click the **Enable all filtered channel(s)** button to send all channels in the **Enabled** column.
3. Click **OK** when you are done.

Browsing Ectane Setup Information

Once you have configured a setup, or opened a setup file, you can browse through all the setup information from the **Setup Information** window.

To access that information window, select **Acquisition Setup>Setup Information** from the **Settings** menu, or click the **Setup information** button. The **Setup Information** window appears.

Figure 3-84 The **Setup Information** window



From this window, you can filter by which category you want to view the setup information:

- Channels
- Frequencies
- Topologies

Note *Information in this window is always the same. Selecting a layout on the left only organizes the information differently, based on what you want to see.*

When you are done browsing the setup information, click **Close.**

Setting Up Processed Channels

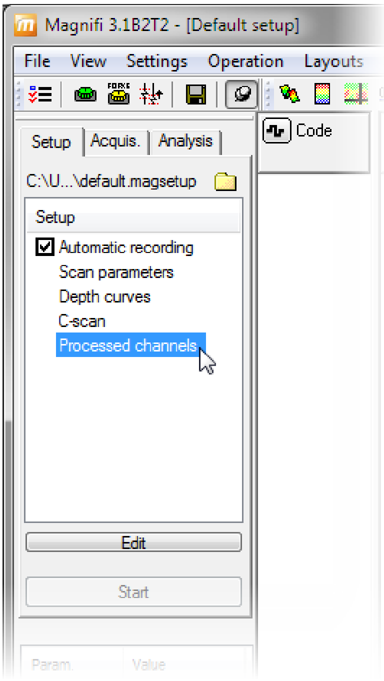
Magnifi offers extremely powerful signal processing capabilities. These capabilities, known as *processed channels*, are in fact a simple, visual programming language aimed specifically at signal processing.

By visually grouping and configuring events (processes), and linking them in a specific order, you can easily apply very complex processing on signals acquired with Ectane instruments.

The following pages explain this programming language. Only the most important events are covered. Once you understand the basic principles of processed channels, you can freely explore the remaining events and learn more, at your own pace.

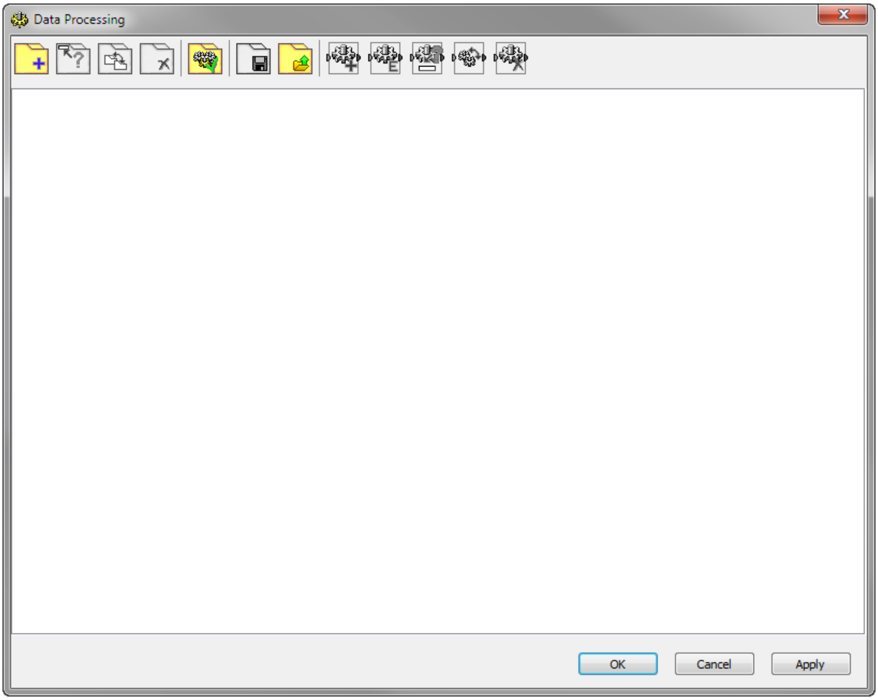
All processed channels are configured from the **Data Processing** window. To access this window, double-click **Processed channels** on the **Setup** tab.

Figure 3-85 Accessing the **Data Processing** window



By double-clicking **Processed channels,** the **Data Processing** window appears.

Figure 3-86 The **Data Processing** window



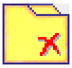
Creating Groups

The first step in creating a channel processing routine (processed channel) is to create a group that will contain all the events (individual processes) applied to acquired signals. To create a group, click **Create a new group** in the **Data Processing** window.

A tab appears in the lower left corner of the **Data Processing** window. This tab is where all events for that group will appear.

Figure 3-87 A group tab at the bottom left of the **Data Processing** window

Deleting Groups



To deleting a group, select the tab of the group that you want to delete and click **Remove current group**.

A dialog box appears, asking you to confirm the deletion. Click **Yes** and the group is deleted.

Duplicating Groups

Duplicating a group is useful when you need to create a new group that is similar, but not identical, to an existing group. By duplicating, you actually create an exact copy of an existing group, a copy that you can modify and rename afterwards.

To duplicate an existing group:

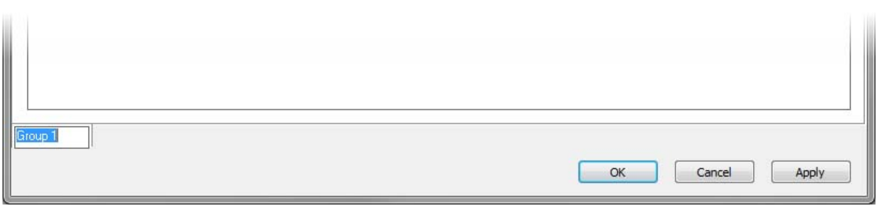
1. In the **Data Processing** window, select the tab of the group that you want to duplicate.

2. Click **Clone current group**. A new group tab appears, displaying the exact content of the previously selected tab.
3. If necessary, rename the group and modify its parameters.

Renaming Groups

By default, the name of a newly created group is **Group *n*,** where “n” is an incremental sequential number. To change that name to a more meaningful or useful name, click **Rename current group.** The name of the tab becomes editable, and you can enter a new name.

Figure 3-88 Renaming a group



Activating/Deactivating Groups

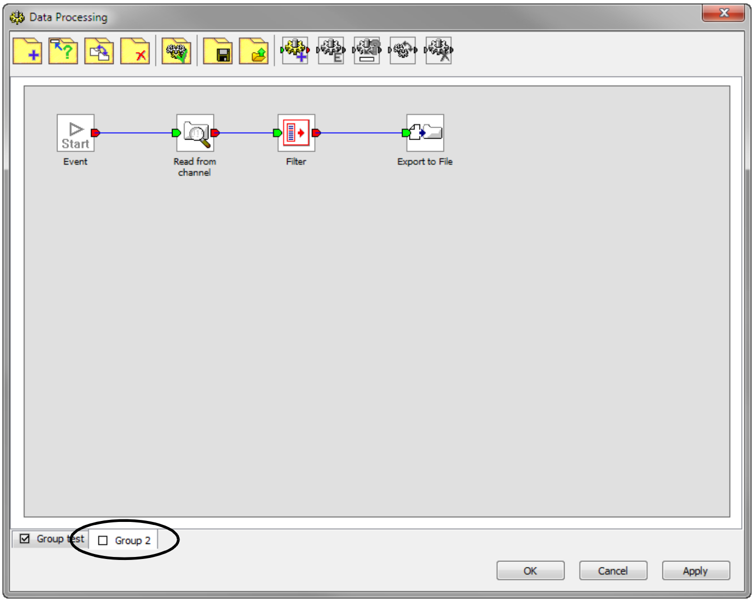
You do not have to remove a group to stop using it. You can simply deactivate it. This keeps the group configuration intact, but makes the group unavailable for signal processing.

To deactivate a group:

1. Select the active group that you want to deactivate.

2. Click **Activate/deactivate the current group**. The mark in the group check box disappears and the back of the tab becomes gray.

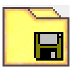
Figure 3-89 A deactivated group



To activate a group, follow the previous procedure, but select a deactivated group first. A mark will appear in the group check box.

Saving Groups

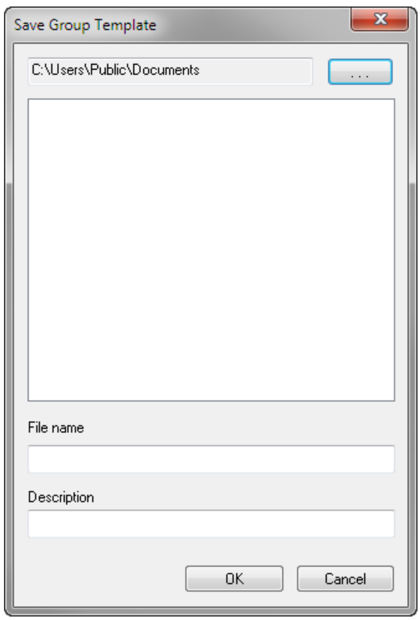
After you have properly configured, or just modified a group of events, you might want to save that group as a template for future use.



To do so:

1. Click **Save current group**. The **Save Group Template** window appears.

Figure 3-90 The **Save Group Template** window



2. Click the "**...**" button. A **Browse to Folder** window appears, where you can choose the location where you want to save the file. Browse to that location and click **OK**. You return to the **Save Group Template** window.
3. Name the file in the **File name** field.
4. If necessary, enter a description of the group in the **Description** field.
5. Click **OK** when you are done. The file is saved in *.xml* format at the location you chose in step 2.

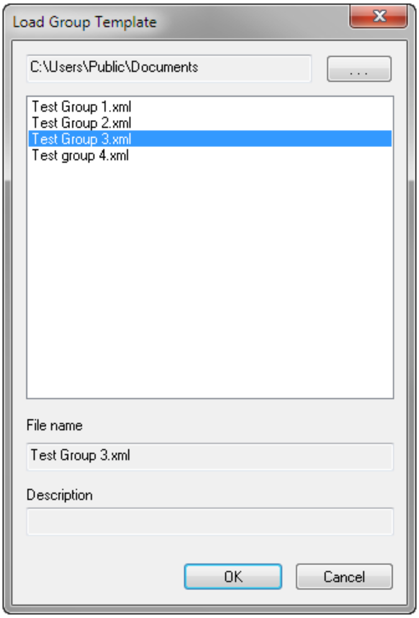
Loading Groups

After you have created and saved a certain number of groups as templates, you can reload them for use at any given time.

To do so:

1. Click **Load group**. The **Load Group Template** window appears.

Figure 3-91 The **Load Group Template** window



2. Select a group from the list and click **OK**.
   The group you selected appears in the **Data Processing** window, ready to use.

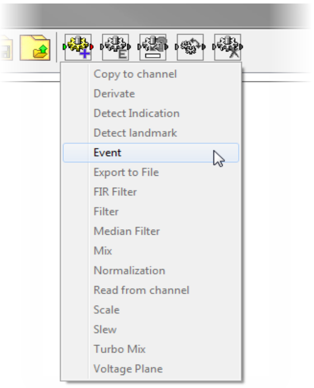
Adding Processing Units

You can only add processing units inside a processing group. Processing units correspond to mathematical actions performed on acquired signals.

To add processing units:

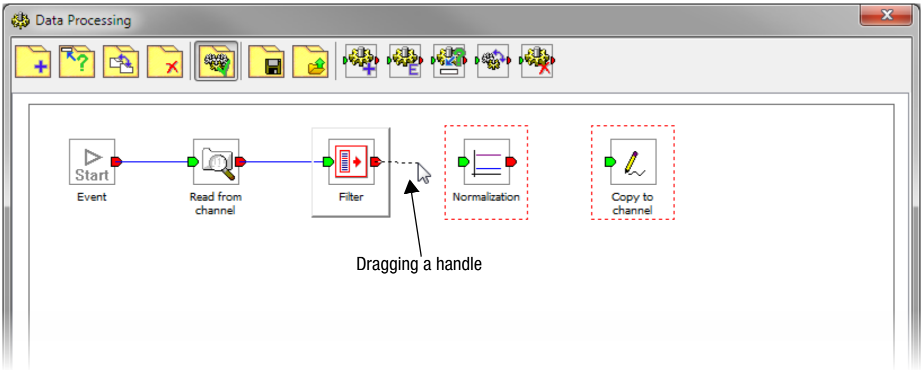
1. Create or select an active group (see "Creating Groups" on page 143).
2. From this group, click the **Create a new unit** button. A menu drops down from the button.

Figure 3-92 Creating a new unit



3. Select the **Event** processing unit (this is the only event available for newly created groups). It acts as a starting point for the signal processing sequence. Once you have added this **Event** processing unit, all other processing units become available.
4. Add a second processing unit. It appears literally on top of the **Event** unit. Drag the new unit at an appropriate location within the group tab.
5. Drag the handle from the **Event** unit to the newly added unit. This symbolizes the logical link between the two units (see Figure 3-93).
6. Repeat steps 4 and 5 until you have created a complete signal processing sequence.

Figure 3-93 Linking processing units



7. After you have finished adding events and linking them, click **OK.** This processed channel will be saved in the setup file.

Deleting Processing Units

To delete a processing unit, simply select it and click **Remove selected unit.**

You can also select the unit and press the DEL key, or right-click the unit that you want to remove, and select **Delete Unit** from the contextual menu that appears.

Editing Processing Units

Save for the **Event** (Start) event, most processing units are editable. To edit a processing unit, simply double-click its icon. The appropriate dialog box appears, where you can modify specific processing parameters. The most often used processing units are explained below.

Note *You can also right-click the unit whose parameters you want to edit, and select **Edit Unit** from the contextual menu that appears.*

Editing the Event Processing Unit

The **Event** processing unit simply acts as a starting point for a signal processing sequence. Nothing about it can be modified. This processing unit must be selected first for the other processing units to become available.

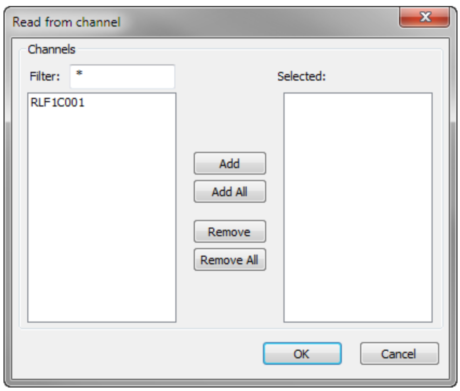
Editing the Read from channel Processing Unit

The **Read from channel** processing unit allows you to choose the channels whose signals will be processed.

To select channels:

1. Double-click the processing unit icon; the **Read from channel** dialog box opens.

Figure 3-94 Configuring the **Read from channel** processing unit



2. Select your channels from the list on the left:
   - To select one channel, click the channel.
   - To select contiguous channels in the list, click one channel, press the SHIFT key, and click the last channel that you want to select. All channels between the first and last selected become selected as well.
   - To select non-contiguous channels in the list, press the CTRL key while clicking on all the channels that you want to select.
3. Click **Add.** All selected channels move to the **Selected** column. If you want to add all channels, simply click **Add All.**
4. Once you are done selecting channels, click **OK.**

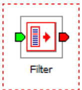
To remove channels from the list of selected channels:

1. Select the channels to remove from the **Selected** column.
2. Click **Remove.** All selected channels are removed from the **Selected** column. If you want to remove all channels, simply click **Remove All.**

Note *If the list contains a large number of channels, you can filter out all irrelevant channels by typing parts of the channel names that you are looking for in the* ***Filter*** *text box. The channels that do not match the entered text are simply not displayed. You can then make your selection from the list of displayed and filtered channels.*

3. Once you are done selecting channels, click **OK.**

Editing the Filter Processing Unit

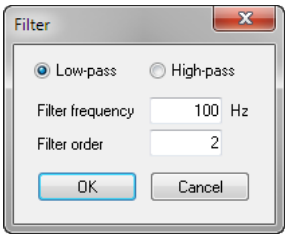


The **Filter** processing unit allows you to filter out certain frequencies from the acquired signals.

To select which frequencies to filter out:

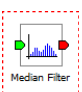
1. Double-click the processing unit icon; the **Filter** dialog box opens.

Figure 3-95 Configuring the **Filter** processing unit



2. Select whether you want to apply a low-pass or high-pass filter.
3. Enter the cutoff frequency that you want to filter out in the **Filter frequency** field.
4. Enter the filter order that you want to filter out in the **Filter order** field.
5. Click **OK.** The **Filter** dialog box disappears, and the **Filter** processing unit is configured.

Editing the Median Filter Processing Unit



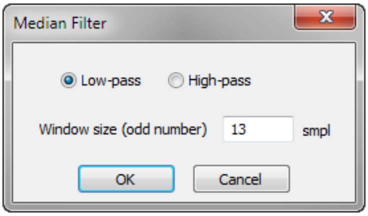
The **Median Filter** processing unit, which is mostly used in ECA inspections, allows you to filter data based on a certain number of samples (a.k.a. “the window”). Using the low-pass filter allows you to keep slow variation signals (lift-off variations or long defects), while using the high-pass filter allows you to keep fast variation signals (sharp defects coming from notches or pits).

Note *The wider the window (high number of samples), the less efficient the high-pass filter will be.*

To configure the median filter processing unit:

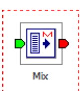
1. Double-click the processing unit icon; the **Median Filter** dialog box opens.

Figure 3-96 Configuring the **Median Filter** processing unit



2. Select whether you want to apply a low-pass or high-pass filter.
3. Enter the number of samples that should constitute the window. The number of samples *must be* an odd number.
4. Click **OK.** The **Median Filter** dialog box disappears, and the **Median Filter** processing unit is configured.

Editing the Mix Processing Unit

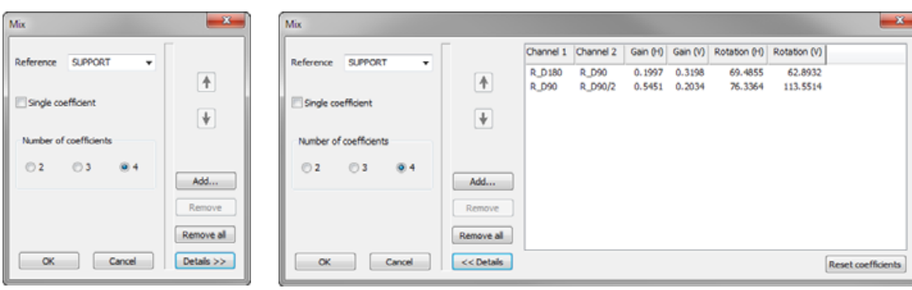


The **Mix** processing unit, which is mostly used for tube inspections, allows you to remove unwanted signals (noise) by subtracting one signal (Channel 2) from another (Channel 1). It can also be used effectively with array probes in specific circumstances.

To configure the **Mix** processing unit:

1. Double-click the processing unit icon; the **Mix** dialog box opens.

Figure 3-97 Configuring the **Mix** processing unit



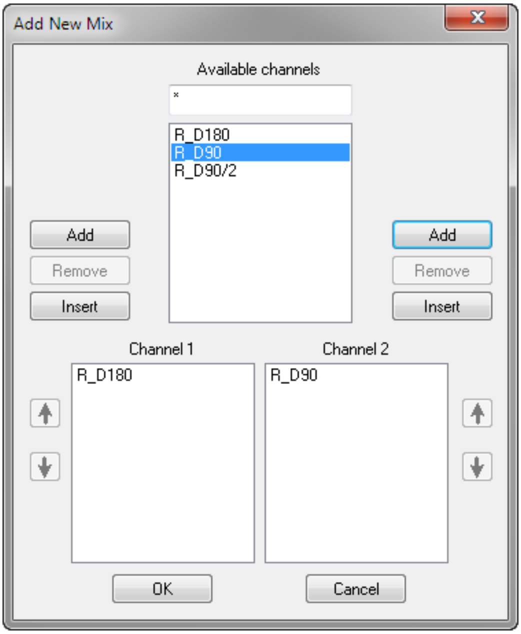
2. In the **Reference** drop-down list, either select an existing reference (created in the **Calibration Points** dialog box; see page 209) or enter the name of a new one (newly entered references will be added to the **Calibration Points** list).

3. In the **Number of coefficients** section, select the number of coefficients applied to the signals.

Note *Eddyfi suggests applying four coefficients. The higher the number of coefficients applied, the more effective the signal removal. You should use less coefficients only if the signals that you want to keep become too distorted.*

4. Click **Add.** The **Add New Mix** dialog box appears.

Figure 3-98 The **Add New Mix** dialog box



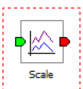
5. From the **Available channels** list, click a channel that you want to use as a base channel.
6. Click the **Add** button on the left-hand side to move the selected channel in the **Channel 1** list.
7. From the **Available channels** list, click the channel that you want to subtract from the channel that you just added to the **Channel 1** list.
8. Click the **Add** button on the right-hand side to move the selected channel in the **Channel 2** list.

9. If necessary, repeat steps 5 to 8 until all the mixes are complete.

Note *Data from channels in the **Channel 2** column are subtracted from the data from the adjacent channels in the **Channel 1** column (i.e., data for the first channel in the **Channel 2** list is subtracted from data from the first channel in the **Channel 1** list; data from the second channel in the **Channel 2** list is subtracted from the data from the second channel in the **Channel 1** list, and so on.) You can reorder the channels in each list by selecting them and using the up- and down-arrows to move them up or down the list.*

10. Click **OK** when you are done. You are brought back to the **Mix** window.
11. Click **OK** again to close the **Mix** window and finish configuring the **Mix** processing unit.

Editing the Scale Processing Unit

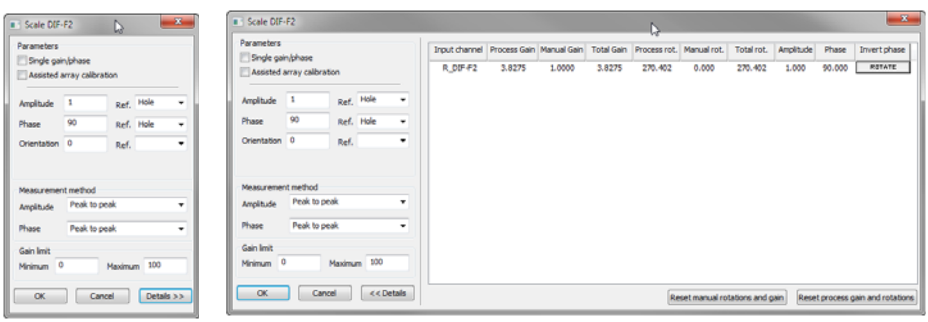


The **Scale** processing unit allows you to adjust the rotation and gain of a channel to bring a specific signal to a predefined angle and amplitude. This process is required when performing a system calibration.

To configure the **Scale** processing unit:

1. Double-click the **Scale** processing unit icon. The **Scale** dialog box appears.

Figure 3-99 Configuring the **Scale** processing unit



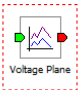
2. If analyzing signals obtained from surface inspection array probes:
   a. Check the **Single gain/phase** box to apply a single gain and phase to all channels. Leave unchecked for individual channel calibration.
   b. Check the **Assisted array calibration** box to help in the determination of the direction of the measurements (vector).

   Otherwise, leave these boxes unchecked. They are not useful for conventional probes.
3. In the **Amplitude, Phase,** and **Orientation** rows, as appropriate, enter the amplitude, phase and/or orientation at which you want to set the signals.

4. On the **Ref.** lists, select the calibration point used to measure the amplitude, phase, and/or orientation.
5. On the **Amplitude** list, select the amplitude measurement method that you want to use:
   - **Absolute:** uses a straight line from the zero position to the main cursor position.
   - **Absolute Horizontal:** uses only the horizontal component, from the imaginary zero level line to the main cursor position.
   - **Absolute Peak:** uses a straight line, from the zero position to the peak value of the data selection.
   - **Absolute Peak Horizontal:** uses only the horizontal component, from the imaginary zero position to the peak value of the data selection.
   - **Absolute Peak Vertical:** uses only the vertical component, from the imaginary zero position to the peak value of the data selection.
   - **Absolute Vertical:** uses only the vertical component, from the imaginary zero level line to the main cursor position.
   - **Ends of the cursor:** uses the data points located at the selection cursor ends' to perform the measurements. Data inside the cursors is ignored.
   - **Horizontal:** uses only the horizontal component.
   - **Peak to peak:** uses a combination of horizontal and vertical components.
   - **PP First transition:** uses the combination of the vertical and horizontal component of the first transition.
   - **Vertical:** uses only the vertical component.
6. On the **Phase** list, select the phase measurement method that you want to use:
   - **Absolute:** uses a straight line from the zero position to the main cursor.
   - **Absolute Peak:** uses a straight line from the zero position to the peak value of the data selection.
   - **Ends of the cursor:** uses the data points located at the selection cursor ends' to perform the measurements. Data inside the cursors is ignored.
   - **Inverted ends of the cursor:** uses the baseline from the top cursor.
   - **Maximum rate:** used to measure the phase between the two points of greatest variation in the Lissajous.
   - **Peak to peak:** used to measure the phase between the two farthest points (extremities of the steepest section) of the Lissajous.
   - **PP First transition:** used to measure the phase between the two farthest points (extremities of the steepest section on the first transition) of the Lissajous.
7. In the **Gain limit** section, enter the minimum and maximum gain that you want to apply when rotating the signal.
8. Click **OK.** The **Scale** processing unit is configured.

Editing the Voltage Plane Processing Unit

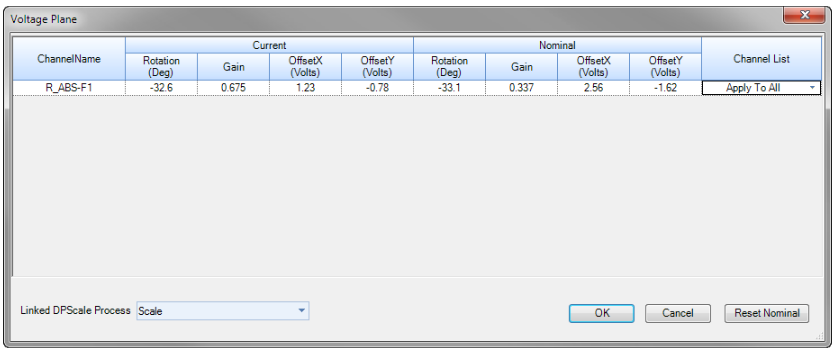
The **Voltage Plane** processing unit is used only with absolute signals in RFT mode. It allows the system to rotate, translate, and scale the signal at the nominal position in the voltage plane view.



To configure the **Voltage Plane** processing unit:

1. Double-click the **Voltage Plane** processing unit icon. The **Voltage Plane** dialog box appears.

Figure 3-100 Configuring the **Voltage Plane** processing unit



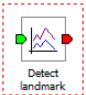
All data shown under **Current** applies to the actual calibration, and all data shown under **Nominal** applies to the recorded nominal point.

2. From the **Linked DPScale Process** drop-down list, select whether you want to link the voltage plane to a scale process.
   Each time a normalization is done in the voltage plane, the same rotation and gain are applied to the linked Scale processing unit.
3. For the previous action to become effective, you must select **Apply To All** in the **Channel List** drop-down list.

Note *If you click **Reset Nominal,** all values under the **Nominal** columns are set to 0.*

Note *If you click the Normalize icon in the voltage plane measurement bar, the **Current** values are updated with new values. If you click the Back to Nominal icon in the voltage plane, the **Current** values are updated with the values displayed in the **Nominal** column. If you click Save nominal, the **Current** values are copied in the **Nominal** sub-columns. For more information on these icons, see* *"Elements of a Voltage Plane View" on page 18.*

Editing the Detect Landmark Processing Unit



The **Detect Landmark** processing unit allows you to detect automatically the landmarks defined in the landmark table (see “Managing Landmarks” on page 180).

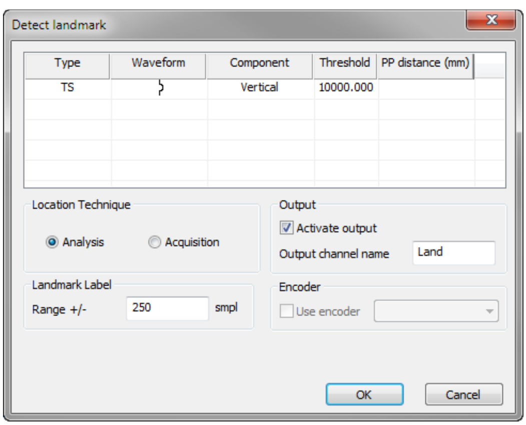
Note *This processing unit is normally added in a separate group to simplify the processed channel. This separate group would include:*

- **Event**
- **Read:** Either from a raw data or a processed data
- **Median filter** (high-pass) or **Derivate:** to center the data on the zero mark
- **Detect Landmark**
- **Copy to Channel:** to have access to a landmark channel

To define this processing unit:

1. Double-click the processing unit icon; the **Detect landmark** dialog box opens.

Figure 3-101 Configuring the **Detect landmark** processing unit



2. All landmark types defined in the landmark table (see page 180) appear in the list. Default values are set to:
   - **Waveform:** positive absolute signal (pictogram).
   - **Component:** Vertical.
   - **Threshold:** 10000 (this value is set so that no detection is performed unless this value is modified, or adjusted automatically through calibration).

   The waveform, component, threshold (and peak-to-peak distance values, for differential waveforms) correspond to the minimum criteria required for automatic detection. You can set each value manually.

3. Select when the detection should be performed:
   - **Analysis:** At the acquisition stop, the first and last landmark are detected. Both these landmarks are used to evaluate the inspection speed, which in turn will allow you to locate intermediate landmarks. This is the preferred method when all defined landmarks are not necessarily found in the tube.
   - **Acquisition:** Landmarks are detected during the acquisition, by using the order and position defined in the landmark table (see page 180). The software estimates positions by using the inspection speed defined in the scan parameters. This is the preferred method when the inspection speed can vary greatly from one tube to another, or even within the same tube, as the detection algorithm accepts significant positioning errors.
4. Define the accepted tolerance (in number of samples) on landmark positioning in the **Landmark Label** field. Higher values can account for greater inspection speed variations.
5. The detection level relative to each landmark can be displayed and adjusted in the Lissajous view of this channel based on the default *Land* channel. To activate this function, check **Activate output** from the **Output** section.
6. If an encoder is present in your setup, you can choose to use it by selecting **Use encoder** in the **Encoder** section. If no encoder is present, this section is deactivated.
7. Click **OK.** The **Detect landmark** dialog box disappears, and the **Detect landmark** processing unit is configured.

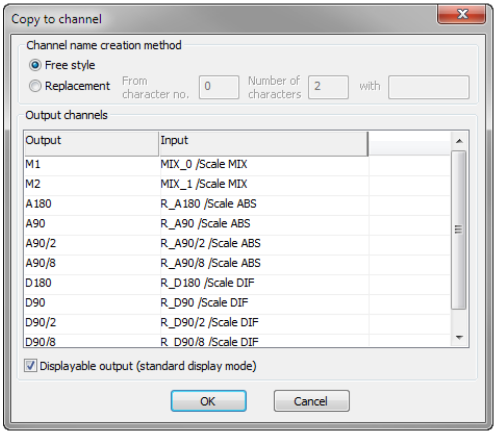
Editing the Copy to channel Processing Unit

The **Copy to channel** processing unit allows you to take all the input channels and output them under a different name after they have been processed. This is the only way to display processed channels in views.

To configure the **Copy to channel** processing unit:

1. Double-click the **Copy to channel** processing unit icon. The **Copy to channel** dialog box appears.

Figure 3-102 Configuring the **Copy to channel** processing unit



2. In the **Channel name creation method** section, select whether you want to name the new output channels manually (**Free style**) or automatically (**Replacement**).
3. If you selected **Free style,** you can then double-click the names in the **Output** column and change them to any name that you want.
4. If you selected **Replacement,** the text boxes on the same line (**From character no., Number of characters,** and **with**) become active.
   - In the **From character no.** box, enter the position of the character *after* which characters will be replaced. If you enter 0, the new characters specified in the **with** field are placed at the beginning of the filename.
   - In the **Number of characters** box, enter the number of characters to replace in the input channel name. If you enter 0, characters will be *added* to the output channel name.
   - In the **with** box, enter the character that will replace or be added to the character(s) that you defined in the two previous boxes.

Table 3-1 Examples of output channel character replacements

Input channel name	From character no.	Number of characters	with	New output name
R_A90/8	0	0	T	TR_A90/8
R_A90/8	1	0	T	RT_A90/8
R_A90/8	0	1	T	T_A90/8

Table 3-1 Examples of output channel character replacements *(continued)*

Input channel name	From character no.	Number of characters	with	New output name
R_A90/8	1	1	T	RTA90/8
R_A90/8	2	2	T	R_T0/8
R_A90/8	2	2	TC	R_TC0/8

5. Click **OK.** The **Copy to channel** processing unit is configured. The output channels are now accessible for further processing under their new name.

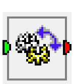
Editing Other Processing Units

The remaining processing units (**Derivate, Detect Indication, Export to File, FIR Filter, Normalization, Slew,** and **Turbo Mix**) are seldom used, or are being overhauled at the time of writing.

Duplicating Processing Units

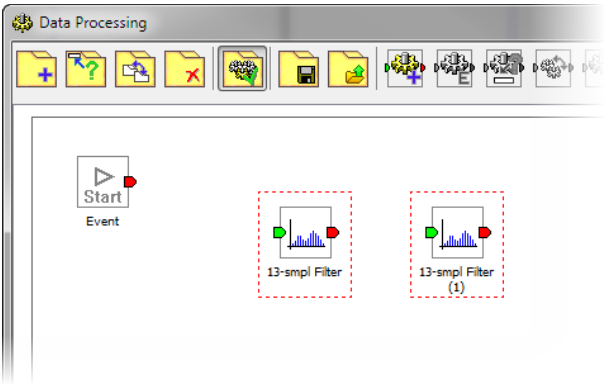
Duplicating allows you to reuse an existing, and already configured, processing unit.

To do so:



1. Select the processing unit that you want to duplicate.
2. Click **Clone selected unit.** A new but identical unit appears in the group, and a number is added, in parentheses, to differentiate it from the original.

Figure 3-103 Duplicating a processing unit



Note *You can also right-click the unit that you want to duplicate, and select* ***Clone Unit*** *from the contextual menu that appears.*

Renaming Processing Units

To rename a processing unit:

1. Select the processing unit that you want to rename and click **Rename selected unit**

   OR

   Click the *name* of the processing unit that you want to change. The name becomes editable.
2. Enter the new name and either press ENTER, or click outside the processing unit. The new name is applied.

Note *You can also right-click the unit that you want to rename, and select* ***Rename Unit*** *from the contextual menu that appears.*

Note *The new name given to a processing unit also appears as the window name when you double-click to edit that processing unit.*

Managing C-scans

C-scans are two-dimensional representations of eddy current data over a scan area. You configure C-scans in a way similar to processed channels, but with the main purpose of modifying the displayed data to better analyze the results.

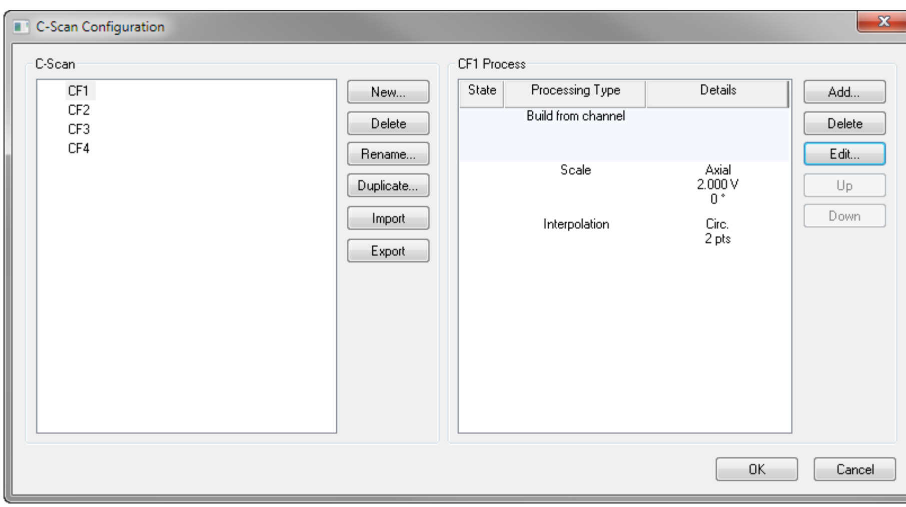
Important C-scan configuration implies that you have an in-depth knowledge and understanding of the way your probe is designed.

Creating C-scans

To create a C-scan:

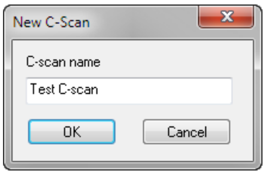
1. With a data file open, select **C-scans** from the **Settings** menu, double-click **C-scan** in the **Setup** tab, or click C-scan from the C-scan toolbar. The **C-scan Configuration** window appears.

Figure 3-104 The **C-scan Configuration** window



2. In the **C-scan** section, click **New.** The **New C-scan** window appears, where you enter the C-scan name in the text field and click **OK.** You return to the **C-scan Configuration** window.

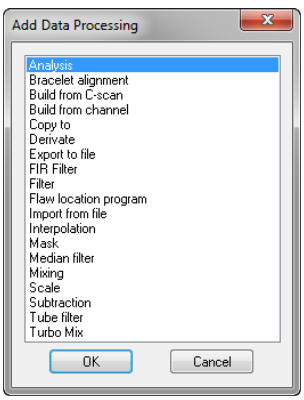
Figure 3-105 The **New C-scan** window



The name of the new C-scan appears selected in the left column (**C-scan**), with a red X beside it (C-scan error), to indicate that it has not been configured yet.

3. In the **Process** section (right), click **Add.** The **Add Data Processing** window appears, where you can select data processing operations.

Figure 3-106 The **Add Data Processing** window



When starting a new C-scan process, you must choose one of the following four processes:

- Build from channel
- Built from C-scan
- Mixing
- Import from File

These processes are explained in the following section (see page 164). Most of the time, you will start a C-scan with the **Built from channel** process. Processes must be configured as you select them from the **Add Data Processing** window.

4. Once you have added all the processes that you want, and properly configured them, click **OK.** The **C-scan Configuration** window disappears and your C-scan is configured.

Editing C-scan Processes

There are many processes available for C-scan processing. Only the most important processes are covered in the following pages. Once you understand the basic principles of C-scan processes, you can freely explore the remaining processes and learn more, at your own pace.

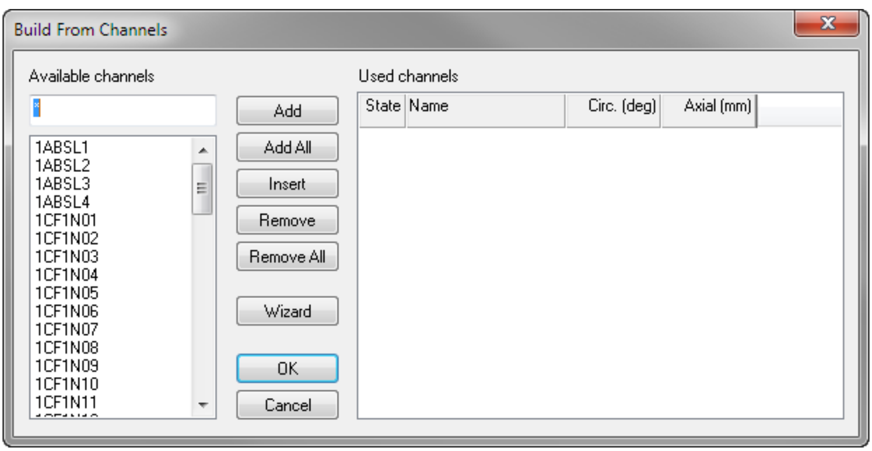
Configuring the Build from channel Process

The **Build from channel** process is usually the first that you add as it is the process by which you select the channels that will be processed and displayed in the C-scan.

To configure the **Build from channel** process:

1. Once you have selected the process from the **Add Data Processing** window (see Figure 3-106), the **Build from Channels** window appears.

Figure 3-107 The **Build from Channels** window



2. From the **Available channels** list, select the channels that you want to process:
   - To select one channel, click the channel.
   - To select contiguous channels in the list, click one channel, press the SHIFT key, and click the last channel that you want to select. All channels between the first and last selected become selected as well.
   - To select non-contiguous channels in the list, press the CTRL key while clicking on all the channels that you want to select.

Important The text box above the list of available channels is in fact a *search box* that allows you to find only the channels that interest you. You can enter part of a channel name and use wildcard signs (*) before and/or after to signify that any other characters can be found before and/or after the part of the channel name that you entered. As you enter text in the search box, only channels meeting the entered criteria will remain.

3. Click **Add** to transfer the selected channels to the **Used channels** list. To add all channels, you do not need to select them first: simply click **Add All.**
4. In the **Used channels** list, enter channel position values in the two columns to the right of the channel name shown.

Note *The columns available to the right of the channel names depend on the type of probe used. These columns accept copy and paste operations, so if you have the proper values already entered in another software (i.e., Microsoft Excel), or even in another C-scan, you can cut these values and paste them directly in the proper columns.*

5. Once you are done entering channel values, click **OK.** The **Build from channels** process is configured.

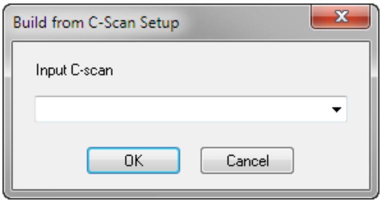
Configuring the Build from C-scan Process

The **Build from C-scan** process can only be the first process in a C-scan. It allows you to start processing from the results of an existing C-scan.

To configure the **Build from C-scan** process:

1. Once you have selected the process from the **Add Data Processing** window (see Figure 3-106), the **Build from C-scan Setup** window appears.

Figure 3-108 The **Build from C-scan** window



2. From the drop-down list, select the C-scan on which you want to base your new C-scan.
3. Click **OK.** The **Build from C-scan** process is configured.

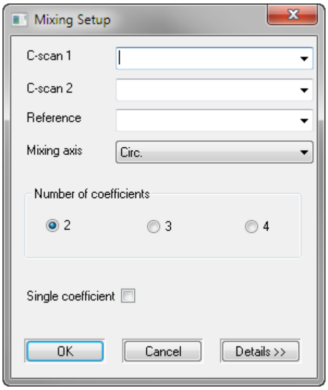
Configuring the Mixing Process

The **Mixing** process, which is mostly used for tube inspections, allows you to remove unwanted signals (noise) by subtracting one C-scan (C-scan 2) from another (C-scan 1). It can also be used effectively with array probes in specific circumstances.

To configure the **Mixing** process:

1. Once you have selected the process from the **Add Data Processing** window (see Figure 3-106), the **Mixing Setup** window appears.

Figure 3-109 The **Mixing Setup** window



2. In the **C-scan 1** drop-down list, select the C-scan from which C-scan 2 will be subtracted.
3. In the **C-scan 2** drop-down list, select the C-scan that will be subtracted from C-scan 1.
4. In the **Reference** drop-down list, either select an existing reference (created in the **Calibration Points** dialog box, [see “Managing Calibration Points” on page 209]) or enter the name of a new one (newly entered references will be added to the **Calibration Points** list).
5. Indicate the axis in which you want to display the data in the **Mixing axis** drop-down list. Usually, this is the scan axis.
6. In the **Number of coefficients** section, select the number of coefficients applied to the signals.

Note *Eddyfi suggests applying four coefficients. The higher the number of coefficients applied, the more effective the signal removal. You should use less coefficients only if the signals that you want to keep become too distorted. The **Single coefficient** check box is normally used with surface array probes to apply the same coefficient to all channels.*

7. Click **OK**. The **Mixing** process is configured.

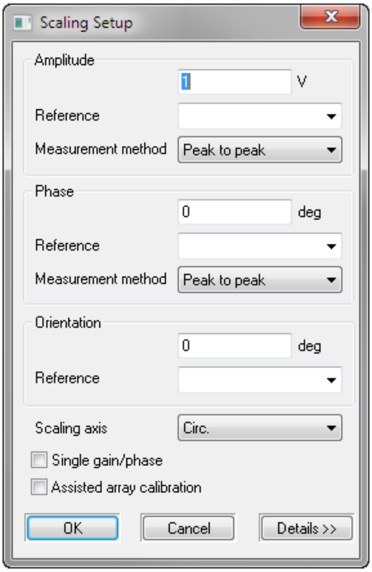
Configuring the Scale Process

The **Scale** process allows you to define the phase angle and amplitude at which a reference signal will be adjusted during the calibration process.

To configure the **Scale** process:

1. Once you have selected the process from the **Add Data Processing** window (see Figure 3-106), the **Scaling Setup** window appears.

Figure 3-110 The **Scaling Setup** window



2. In the **Amplitude** section:
   - **a** Enter the amplitude, in volts, at which you want to set the reference signal.
   - **b** In the **Reference** drop-down list, select the calibration point on which to base this modified amplitude.
   - **c** In the **Measurement method** drop-down list, select the measurement method that you want to use (see page 155).
3. In the **Phase** section:
   - **a** Enter the angle, in degrees, at which you want to set the reference signal.
   - **b** In the **Reference** drop-down list, select the calibration point on which to base this modified phase.
   - **c** In the **Measurement method** drop-down list, select the measurement method that you want to use (see page 155).

4. In the **Orientation** section (optional):
   a Enter the approximate angle, in degrees, at which the reference signal should be located.
   b In the **Reference** drop-down list, select the calibration point on which to base this modified orientation.
5. In the **Scaling axis** drop-down list, select the axis along which the scaling will be performed: **Circ.** or **Axial,** or **X** or **Y**, depending on scan parameters.
6. If analyzing signals obtained from surface inspection arrays:
   a Check the **Single gain/phase** box to apply a single gain and phase to all channels, or leave the box unchecked to allow individual adjustment of each probe element.
   b Check the **Assisted array calibration** box to help in the determination of the direction of the measurements (vector).
7. Click **OK.** The **Scale** process is configured.

Configuring the Median Filter Process

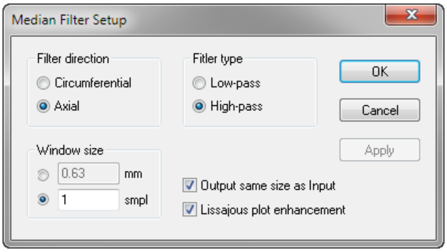
The **Median filter** process, which is mostly used in ECA inspections, is a statistical filter based on the median value calculated on a certain number of samples (a.k.a. “the window”). It can behave as either a low- or high-pass filter with the particularity that it does not create signal distortion.

Note *The wider the window (high number of samples), the less efficient the high-pass filter will be.*

To configure the **Median Filter** process:

1. Once you have selected the process from the **Add Data Processing** window (see Figure 3-106), the **Median Filter Setup** window appears.

Figure 3-111 The **Median Filter Setup** window



2. In the **Filter direction** section, select the direction in which the filter will perform: **Circumferential** or **Axial,** or **X** or **Y**, depending on scan parameters.

3. In the **Filter type** section, select whether you want to apply a low-pass or high-pass filter. Using the low-pass filter allows you to keep slow variation signals (long defects, lift-off), while using the high-pass filter allows you to keep fast variation signals (sharp defects such as cracks, notches, and pits).
4. In the **Window size** section, enter the number of samples that should constitute the window *or* the size of the window in mm or degrees. The number of samples must be an odd number.
5. If you want the median filter output to be the same size as the data input, select **Output same size as Input.** This adds points at the beginning and end of the C-scan.
6. If you do not want to enhance the quality of Lissajous plots, clear the **Lissajous plot enhancement** check box.
   It is selected by default.
7. Click **OK.** The **Median Filter Setup** window disappears, and the **Median Filter** process is configured.

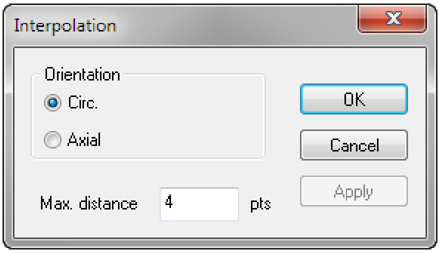
Configuring the Interpolation Process

The interpolation process allows you to improve the look of the C-scans by "smoothing" the details on screen without losing data integrity.

To configure the **Interpolation** process:

1. Once you have selected the process from the **Add Data Processing** window (see Figure 3-106), the **Interpolation** window appears.

Figure 3-112 The **Interpolation** window



2. In the **Orientation** section, select the direction along which data points will be interpolated: **Circ.** or **Axial,** or **X** or **Y**, depending on scan parameters.

Important You also need to reopen the scan parameters (see "Configuring Scan Parameters" on page 104) to reduce the C-scan grid resolution for the same axis on which the interpolation process was set, i.e., an initial Circ. resolution of six degrees. If the resolution is reduced to two degrees, two interpolation points will be added between real data points.

3. In the **Max. distance** text box, enter the number of points that will be interpolated in the C-scan. The higher the number of interpolation points, the "smoother" the displayed results, but also the greater the risk of losing sight of potentially meaningful data points.
4. Click **OK.** The **Interpolation** window disappears, and the **Interpolation** process is configured.

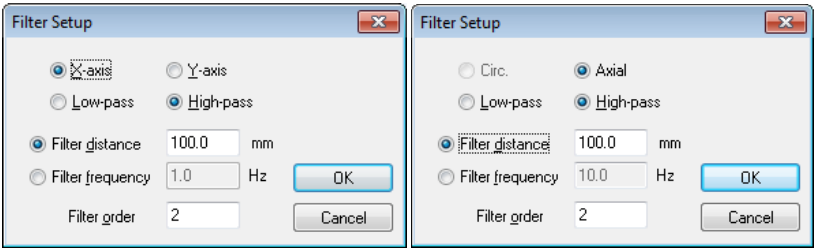
Configuring the Filter Process

The **Filter** process allows you to remove certain signal frequencies (high or low) in a displayed data.

To configure the **Filter** process:

1. Once you have selected the process from the **Add Data Processing** window (see Figure 3-106), the **Filter Setup** window appears.

Figure 3-113 Filter Setup window



2. Select whether you want to apply your filter to **X** or **Y** (surface geometry scans), or to **Axial** (tube geometry scans).
3. Select whether you want it to be high-pass or low-pass. Using a low-pass filter allows you to keep slow variation signals (long defects, lift-off), while using the high-pass filter allows you to keep fast variation signals (sharp defects such as cracks, notches, and pits).
4. Select whether you want to filter based on distance or frequency. If you select **Filter distance,** enter the distance in the **mm** field. If you select **Filter frequency,** enter the cut-off frequency that you want to filter out in the **Hz** field.
5. In the **Filter order** field, enter the filter order that you want to filter out.
6. Click **OK.** The **Filter Setup** dialog box disappears, and the **Filter** process is configured.

Deleting C-scan Processes

To delete a process, select it from the **Process** list and click **Delete.** The selected process disappears from the list of C-scan processes.

Note *Processes that must be present to start a C-scan process (**Build from channels, Build from C-scan, Mixing,** etc.) cannot be deleted.*

Ordering C-scan Processes

To change the position of a process in the list of C-scan processes, select the process to move and click **Up** or **Down** until it reaches the position that you want in the list.

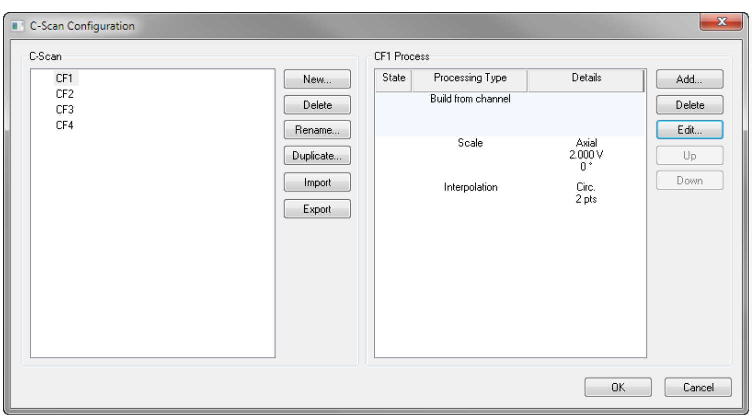
Note *Processes that must be present to start a C-scan process (**Build from channels, Build from C-scan, Mixing,** etc.) cannot be moved to another position in the list.*

Deleting C-scans

To delete an entire C-scan:

1. With a data file open, either select **C-scans** from the **Settings** menu, or double-click **C-scan** in the **Setup** tab. The **C-scan Configuration** window appears.

Figure 3-114 The **C-scan Configuration** window



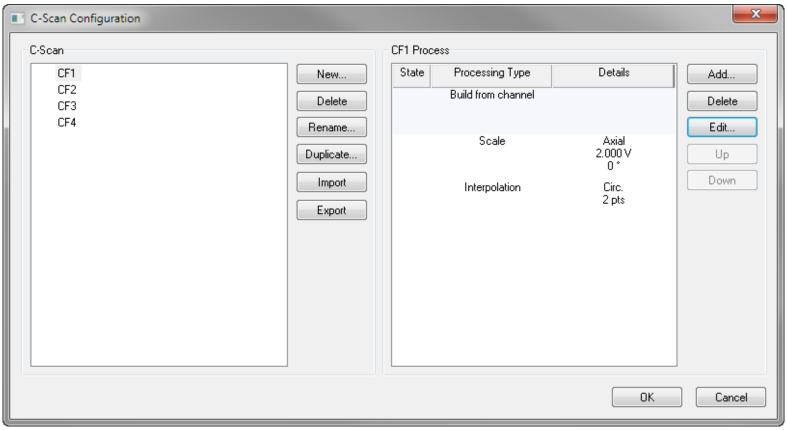
2. In the **C-scan** list on the left, select the C-scan to delete and click **Delete.** The C-scan is removed from the list.

Duplicating C-scans

To duplicate a C-scan:

1. With a data file open, either select **C-scans** from the **Settings** menu, or double-click **C-scan** in the **Setup** tab. The **C-scan Configuration** window appears.

Figure 3-115 The **C-scan Configuration** window



2. In the **C-scan** list on the left, select the C-scan to duplicate and click **Duplicate.** A new C-scan is added to the list with the name of the original to which a "_*n*" has been added, where "*n*" is an incremental variable that starts at the lowest *available* integer.

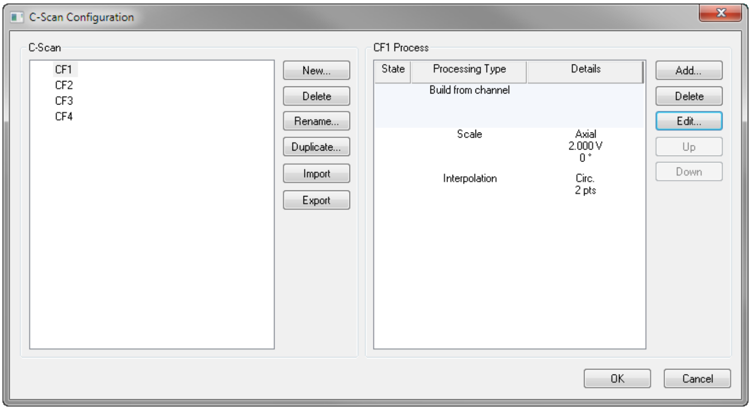
   For example, in Figure 3-115, if duplicating the **CF1** C-scan, a **CF1_1** C-scan is automatically added. If you select **CF1** to duplicate it once more, **CF1_2** is created. If **CF1_1** is subsequently deleted, and **CF1** is duplicated again, it will create **CF1_1,** and not **CF1_3.**

Renaming C-scans

To rename a C-scan:

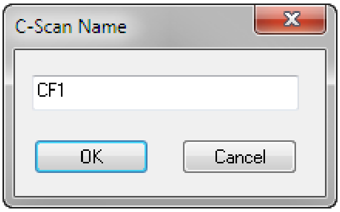
1. With a data file open, either select **C-scans** from the **Settings** menu, or double-click **C-scan** in the **Setup** tab. The **C-scan Configuration** window appears.

Figure 3-116 The **C-scan Configuration** window



2. In the **C-scan** list on the left, select the C-scan to rename and click **Rename.** The **C-scan Name** window appears with the previous name appearing in the text box.

Figure 3-117 Renaming a C-scan



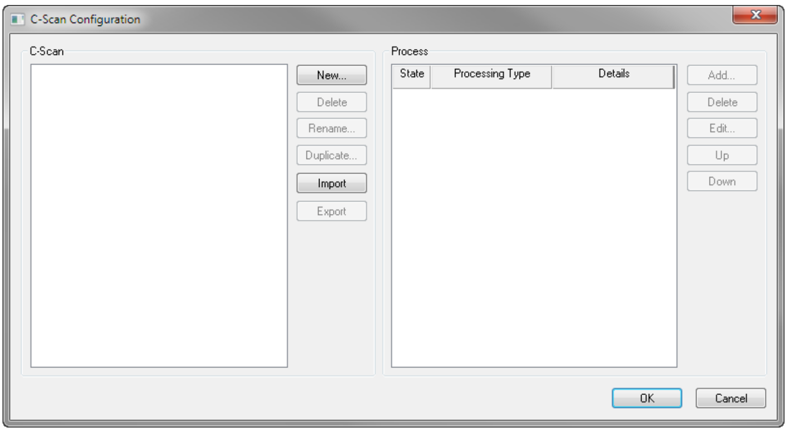
3. Enter the new name in the text box and click **OK.** The **C-scan Name** window disappears and you return to the **C-scan Configuration** window, where the C-scan now appears under its new name.

Importing C-scans

To import C-scans:

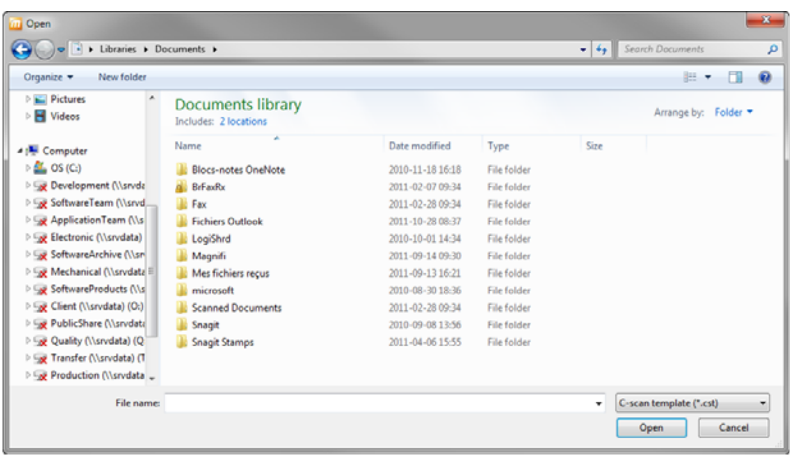
1. Select **C-scans** from the **Settings** menu, or double-click **C-scan** in the **Setup** tab. The **C-scan Configuration** window appears.

Figure 3-118 The **C-scan Configuration** window



2. Click **Import.** A standard **Open** dialog box appears.

Figure 3-119 Importing a C-scan



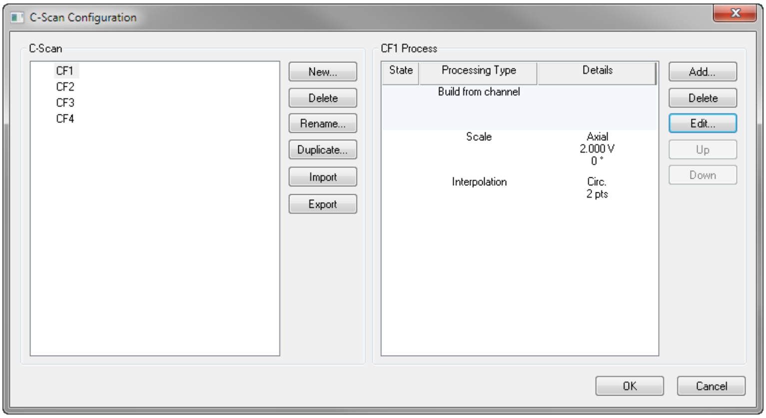
3. Browse to the directory where the C-scan template file (*.*cst*) is located.
4. Select the file and click **Open.** You return to the **C-scan Configuration** window, and the C-scan is now available.

Exporting C-scans

To export C-scans:

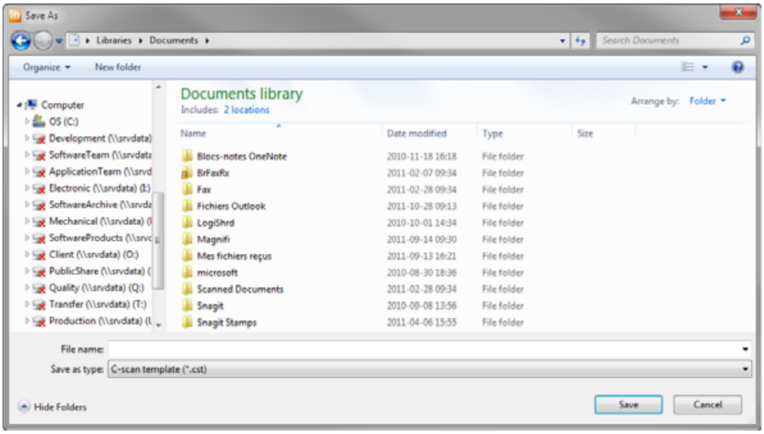
1. With a data file open, either select **C-scans** from the **Settings** menu, or double-click **C-scan** in the **Setup** tab. The **C-scan Configuration** window appears.

Figure 3-120 The **C-scan Configuration** window



2. Select the C-scan to export.
3. Click **Export.** A standard **Save As** dialog box appears.

Figure 3-121 Exporting a C-scan



4. Browse to the directory where you want to export the C-scan template file (*.*cst*) and click **Save.** The *.*cst* file is saved in the selected directory.

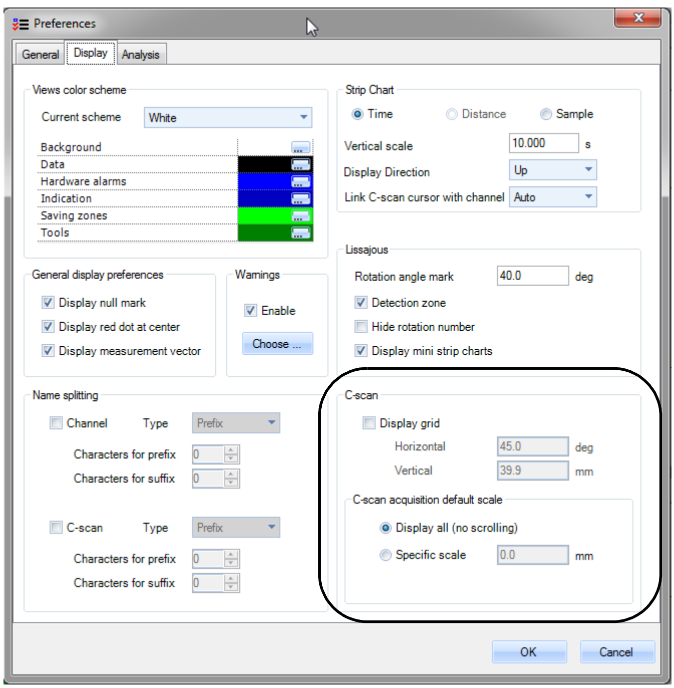
Configuring C-scan Views

When displaying a C-scan view in a layout, you can choose to overlay a grid on top of the C-scan, and you can also decide of the scale applied to the C-scan. Displaying a grid can help you separate channels more easily, among other things, and changing the scale can modify the amount of scrolling that you do when viewing a C-scan.

To configure C-scan views:

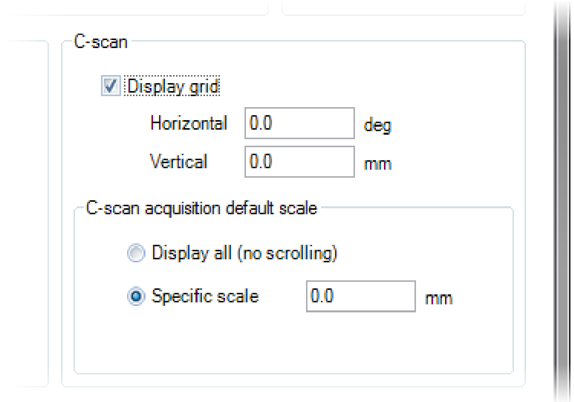
1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. Click the **Display** tab.

Figure 3-122 The **Display** tab



3. In the **C-scan** section, check the **Display grid** box if you want to display a grid. Once you check this box, the **Vertical** and **Horizontal** text boxes become active.

Figure 3-123 The **C-scan** section



4. Enter the values for the horizontal and vertical size of the grid that you want to overlay on top of the C-scan.
5. In the **C-scan acquisition default scale** subsection, choose the scale that you want to apply to the C-scan:
   - **Display all (no scrolling):** the entire C-scan data is displayed in the view.
   - **Specific scale:** allows you to enter the exact size that you want to give to the C-scan.
6. Once you are done, click **OK.** The C-scan views will be properly configured.

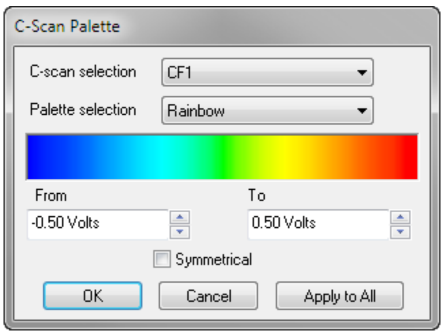
Changing C-scan Palettes in Use

C-scan data is displayed using a color palette. Each C-scan can have its own color palette.

To change the color palette for a specific C-scan:

1. Select **C-scan Palette** from the **Operation** menu, or click **C-scan palette** from the C-scan toolbar. The C-scan Palette window appears.

Figure 3-124 The **C-scan Palette** window



2. From the **C-scan selection** drop-down list, select the C-scan to which you want to attribute a color palette.
3. From the **Palette selection** drop-down list, select one of the color palettes offered. The selected palette appears in the box underneath the drop-down list and is automatically applied to all the C-scan views corresponding to the C-scan selected in step 2.
4. You can change the limit values (in volts) for either extremities of the color palette using the **From** and **To** text boxes. To keep these values symmetrical on both sides of the 0, check the **Symmetrical** box. This will ensure that when you change either value, the other value follows suit symmetrically.
5. If you want to apply the selected color palette to all C-scans, click **Apply to All.**
6. When you are done assigning color palettes to C-scans, click **OK.**

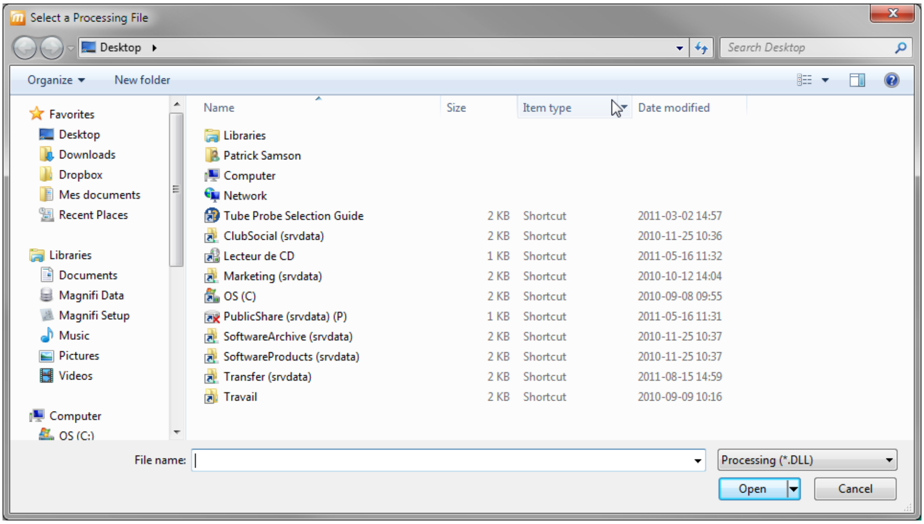
Registering Processing

Eddyfi can create custom C-scan processes not delivered with the standard versions of Magnifi. To add theses processes for use in Magnifi, you have to register them first.

To do so:

1. From the **Special** menu in Magnifi, select **Register Processing.** The **Select a Processing File** window appears.

Figure 3-125 The **Select a Processing File** window



2. Find the processing file (*.*dll*) that you need and double-click it. You return to Magnifi as if nothing happened, but the processing file that you just selected is now registered.
3. Restart Magnifi.

From now on, the processing file that you registered will be available from the **Add Data Processing** window.

Managing Landmarks

In Magnifi, you can manage landmarks manually or automatically. The following pages explain how to do both.

Below is the list of operations to perform when managing landmarks

- Set the direction of the displayed data (for more information, see “Configuring General Strip Chart Parameters” on page 44.)
- Build a landmark table
- Set the landmark positioning options
- Manually place landmarks

Building Landmark Tables

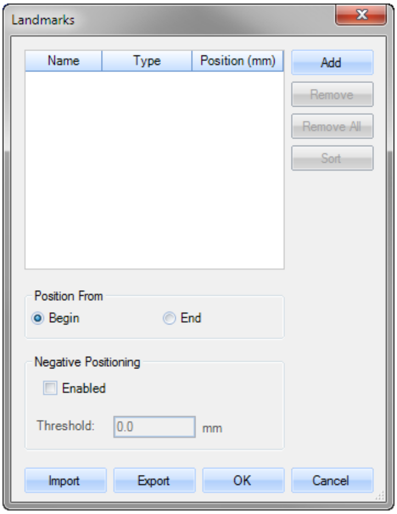
Magnifi allows you to create landmarks in software, provided that you know the exact position of each of these landmarks.

Note *If you do not know the exact position of the landmarks, it is better to simply enter “begin” and “end” landmarks.*

To do so:

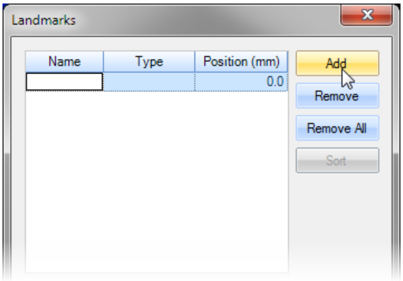
1. From the **Settings** menu, select **Landmarks.** The **Landmarks** window appears.

Figure 3-126 The **Landmarks** window



2. Click **Add.** An empty line is added to the list on the left.

Figure 3-127 Adding a landmark



3. Enter the name of the landmark in the **Name** column and press TAB. The **Type** field is now selected.
4. Enter the type of landmark in the **Type** column (tubesheet, support plate, baffles, etc.) The types entered here will be used elsewhere for automatic detection (see "Editing the Detect Landmark Processing Unit" on page 157).
5. Press TAB. The **Position (mm)** field is now selected. Enter the position of the landmark.

Note *The distance unit used in this window depends on the one selected in the* ***Preferences*** *window. See* *"Setting Measurement Units" on page 73.*

6. In the **Position From** section, select whether you create your landmarks from the beginning or the end of the tube.

Important **Keep in mind that the beginning and end positions apply to *the acquisition and not the tube.* Usually, acquisitions start at the end of a tube, and end at the beginning of the tube as you pull the probe *towards* you. Landmarks must be entered in the order in which they will be detected.**

Normally, the position of the defect signal is given based on the distance *added* to a previously known landmark. However, depending on the situation, it might be preferable to subtract a distance from a *following* detected landmark. This is known as *negative positioning*.

7. If you want to use negative positioning, check the **Enabled** box in the **Negative Positioning** section. This activates a threshold value box where you indicate the threshold at which negative positioning becomes effective.

To better understand these principles, see the following figures.

Figure 3-128 Positioning concept 1 (Position from beginning, positive positioning, display direction down)

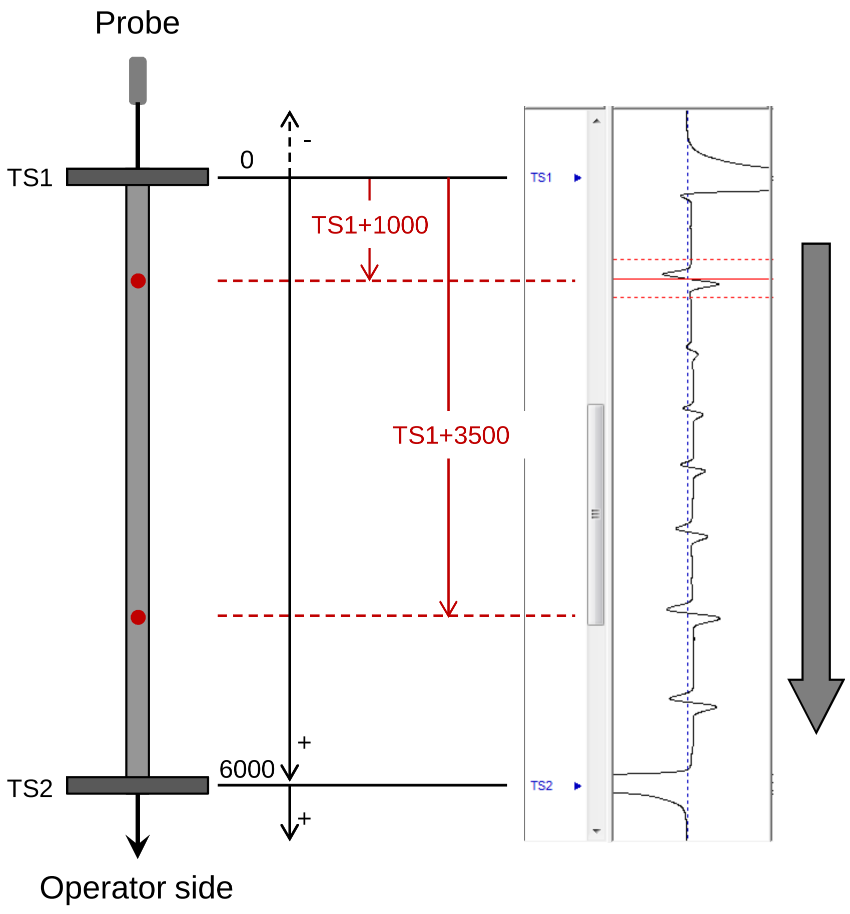
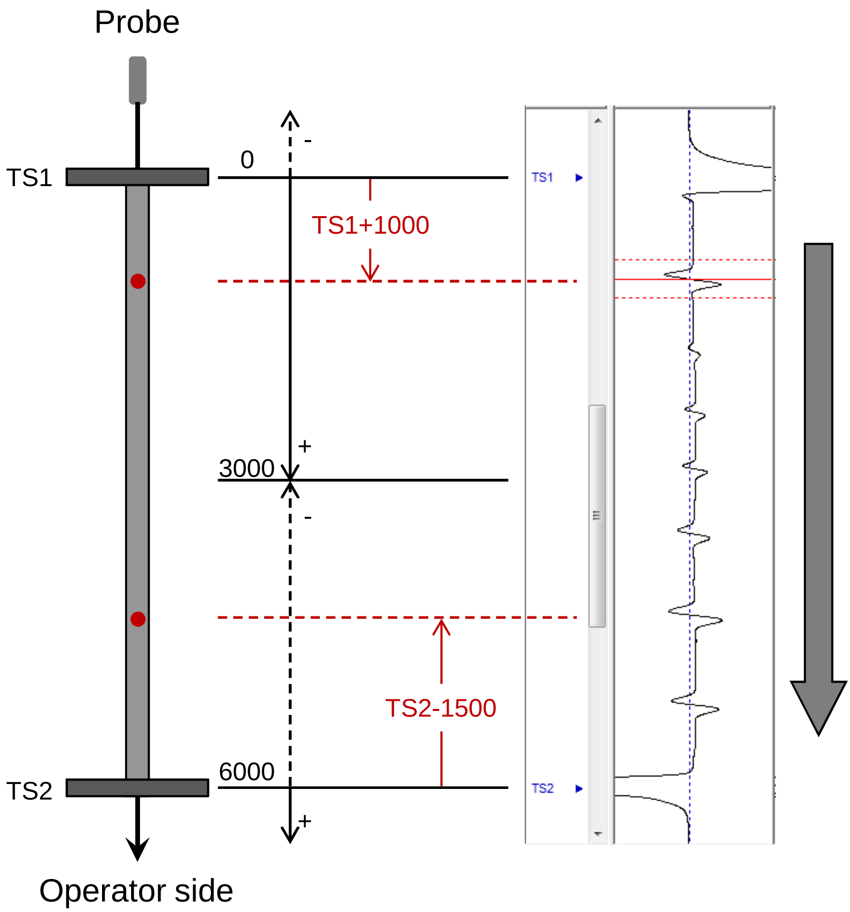


Figure 3-129 Positioning concept 2 (Position from beginning, negative positioning of 3000, display direction down)



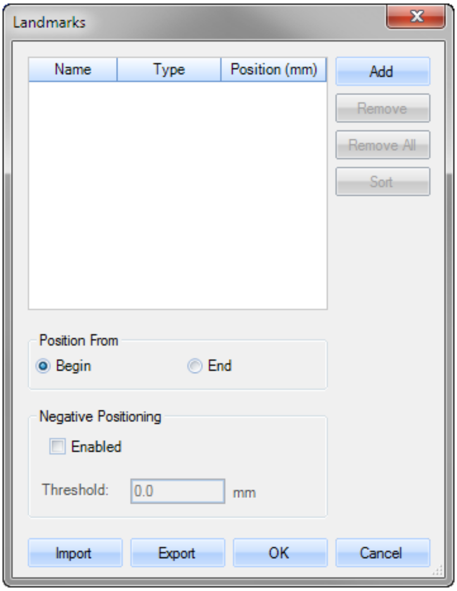
8. Once you are done selecting the positioning, you can sort your landmarks by clicking **Sort.** This allows you to continue modifying the landmark table. Otherwise, the landmark table will be automatically sorted when you click **OK.**
9. You can also remove landmarks by selecting them from the list and clicking **Remove.** You can also start your landmark table anew by clicking **Remove All.**
10. When you are done setting up your landmark table, click **OK.**

Importing Landmarks

To do so:

1. From the **Settings** menu, select **Landmarks.** The **Landmarks** window appears.

Figure 3-130 The **Landmarks** window



2. Click **Import.** A standard **Open** window appears.
3. Browse to find the appropriate landmark file (in **.xml* format) and click **Open,** or double-click the file icon. Landmarks are imported in the **Landmarks** window.

Exporting Landmarks

To do so:

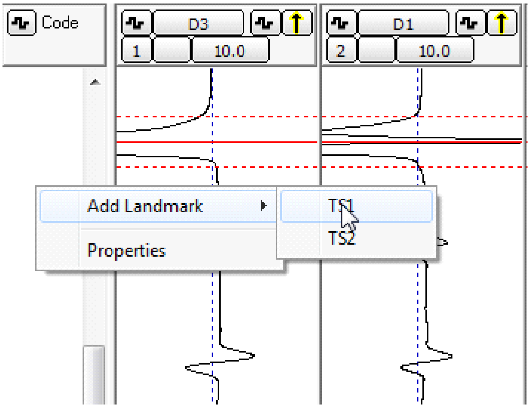
1. From the **Settings** menu, select **Landmarks.** The **Landmarks** window appears.
2. Click **Export.** A standard **Save As** window appears.
3. Browse to find the appropriate location for the landmark file and click **Save**. Landmarks are exported (in **.xml* format) to the selected location.

Adding Landmarks Manually

To add landmarks manually:

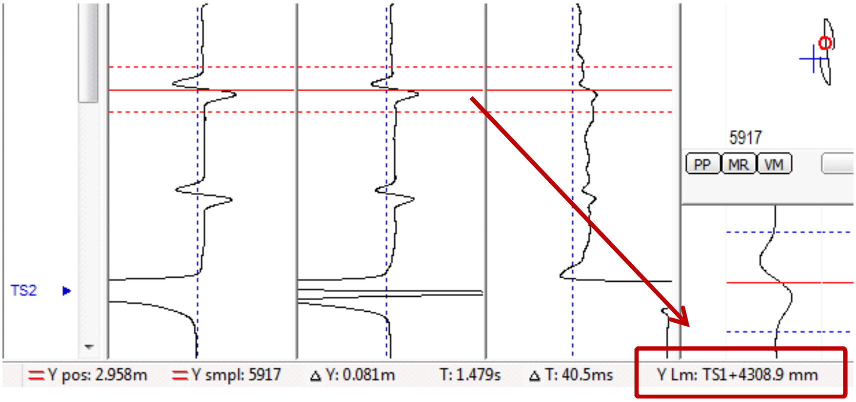
1. Place the data cursor at the first landmark.
2. Right-click in the code window and select the landmark to add, or press F9 to add the first landmark in the list. The list of available landmarks is the one created in the landmark table (see “Building Landmark Tables” on page 180.)

Figure 3-131 Selecting a landmark to add



Once set, the position relative to the landmark is given at the bottom of the screen.

Figure 3-132 Manually-added landmark position



Managing Materials

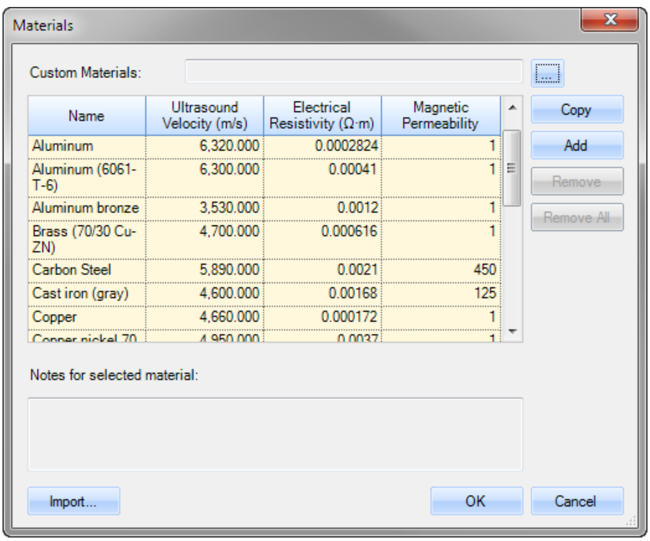
Magnifi features a material manager that contains the basic properties necessary to perform electromagnetic and IRIS inspections.

Materials are managed in the **Materials** dialog box. From there, you can:

- Add custom materials from files
- Create new materials
- Copy existing materials (to modify certain parameters)
- Remove custom materials
- Add notes to materials
- Import material files

To open the **Materials** dialog box, on the **Settings** menu, click **Materials**.

Figure 3-133 Materials dialog box



Adding New Materials

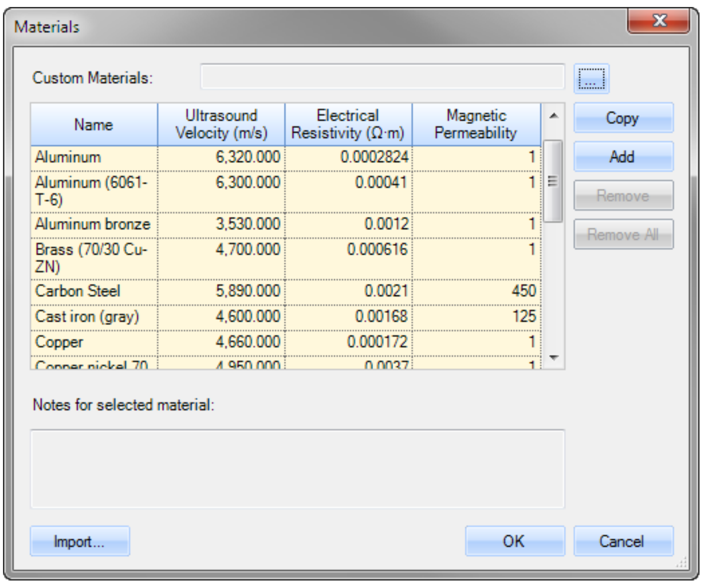
Before adding a new material, you have to know the three most important characteristics of this material, from a testing standpoint:

- the speed of ultrasound in the material (in m/s)
- the electrical resistivity of the material (in $\Omega \cdot \text{m}$)
- the magnetic permeability of the material

To add a new material to the default list:

1. On the **Settings** menu, click **Materials**. The **Materials** dialog box appears.

Figure 3-134 Materials dialog



2. Click **Add.** A new empty line is added at the bottom of the list.
3. Enter the name of the material in the **Name** column.
4. Enter the ultrasound velocity in the material in the **Ultrasound Velocity (m/s)** column.
5. Enter the material's electrical resistivity in the **Electrical Resistivity ($\Omega \cdot \text{m}$)** column.
6. Enter the material's magnetic permeability in the **Magnetic Permeability** column.
7. If necessary, you can enter notes relative to this newly created material.
8. Click **OK.** The new material is saved.

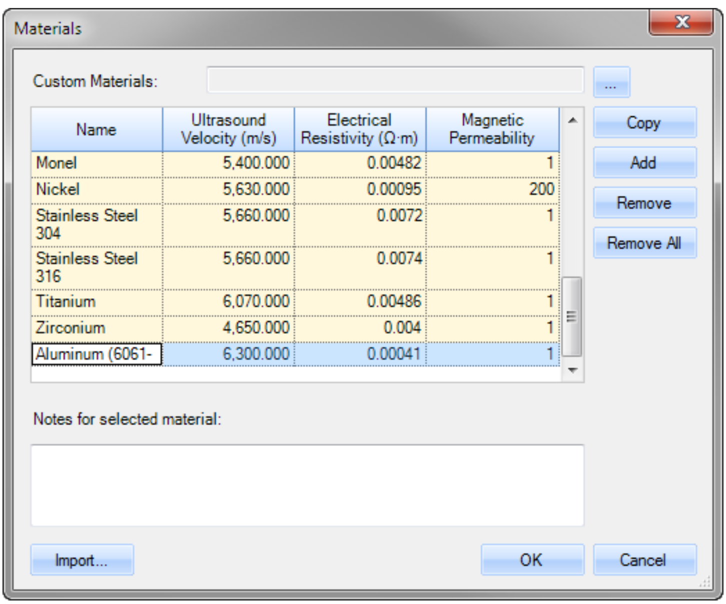
Copying Existing Materials

Copying an existing material is generally used when that existing material already has most the characteristics of a new material that you want to add. This speeds up the process because you do not have to re-enter all the parameters for the new material.

To copy an existing material:

1. On the **Settings** menu, click **Materials**. The **Materials** dialog box appears.
2. In the list of existing materials, select the one that you want to copy.
3. Click **Copy**. A new line is created at the bottom of the list. It contains the same information as the one that was selected before you clicked **Copy.**

Figure 3-135 Copied material



4. Modify the given information as needed (material name, ultrasound velocity, etc.)
5. Enter notes on the new material if necessary.
6. Click **OK.** The new material is created.

Removing Existing Materials

Note *You cannot remove materials from the default list.*

To remove an existing material:

1. On the **Settings** menu, click **Materials**. The **Materials** dialog box appears.
2. In the list of existing materials, select the one that you want to remove.
3. Click **Remove.** The selected material is removed from the list.

Note *You can also quickly remove all added materials by clicking **Remove All.** This will not remove the default materials list.*

Adding Notes to Materials

Note *You cannot add notes to default materials.*

To do so:

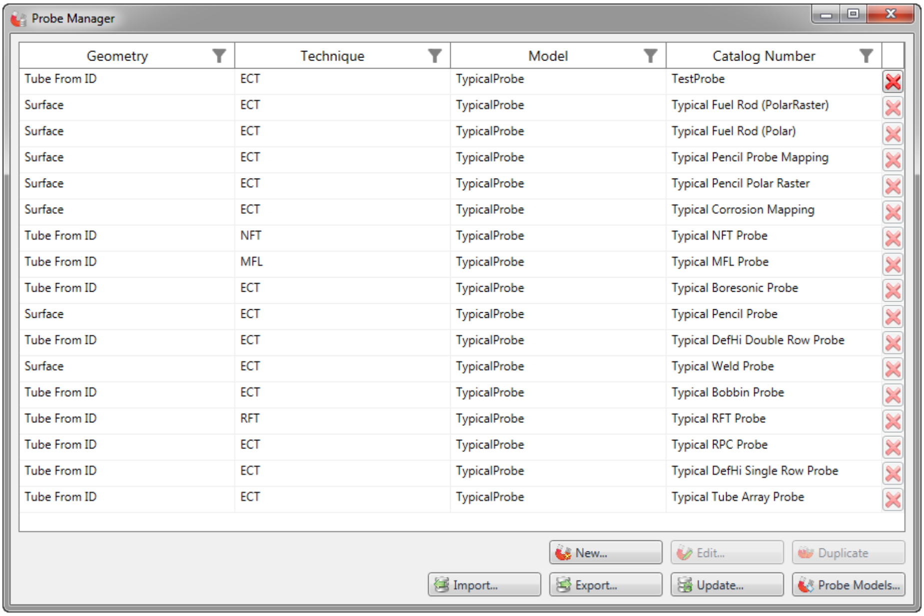
1. On the **Settings** menu, click **Materials**. The **Materials** dialog box appears.
2. In the list of existing materials, select the one to which you want to add notes.
3. Enter your notes in the **Notes for selected material** text box.
4. Click **OK.** Your notes are saved.

Managing Probes

Magnifi comes with an official Eddyfi probe database containing all Eddyfi probes (**Settings>Probe Database**). This database helps speed up configuration operations, especially in the Setup Wizard (see “Preparing Acquisition Setups with the Wizard” on page 65).

These probes cannot be removed from the database. However, there are still certain operations that can be performed on new “user” probes. These operations are explained in the following pages.

Figure 3-136 Probe Manager window



Geometry	Technique	Model	Catalog Number
Tube From ID	ECT	TypicalProbe	TestProbe
Surface	ECT	TypicalProbe	Typical Fuel Rod (PolarRaster)
Surface	ECT	TypicalProbe	Typical Fuel Rod (Polar)
Surface	ECT	TypicalProbe	Typical Pencil Probe Mapping
Surface	ECT	TypicalProbe	Typical Pencil Polar Raster
Surface	ECT	TypicalProbe	Typical Corrosion Mapping
Tube From ID	NFT	TypicalProbe	Typical NFT Probe
Tube From ID	MFL	TypicalProbe	Typical MFL Probe
Tube From ID	ECT	TypicalProbe	Typical Boresonic Probe
Surface	ECT	TypicalProbe	Typical Pencil Probe
Tube From ID	ECT	TypicalProbe	Typical DefHi Double Row Probe
Surface	ECT	TypicalProbe	Typical Weld Probe
Tube From ID	ECT	TypicalProbe	Typical Bobbin Probe
Tube From ID	RFT	TypicalProbe	Typical RFT Probe
Tube From ID	ECT	TypicalProbe	Typical RPC Probe
Tube From ID	ECT	TypicalProbe	Typical DefHi Single Row Probe
Tube From ID	ECT	TypicalProbe	Typical Tube Array Probe

Sorting the Displayed Database

You can sort the probe database by any of its four columns. To do so, simply click in the column title. An arrowhead appears in the column title, indicating whether the column is sorted in alphabetical order (“up” arrow on the left) or in reverse alphabetical order (“down” arrow on the left).

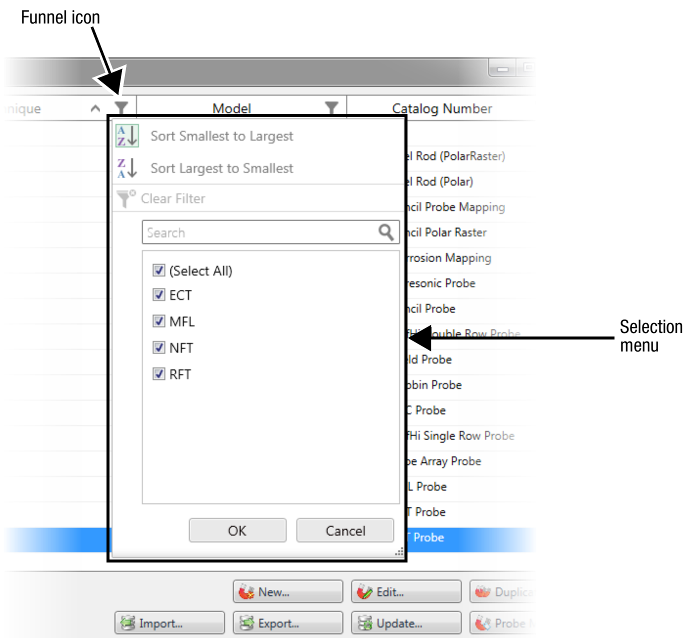
Selecting the Content of a Column

In the **Probe Manager** window, not only can you sort its content, you can also filter out the information that you do not want or need to see.

To do so:

1. From any column in the **Probe Manager,** click the funnel icon. A selection menu appears.

Figure 3-137 Selection menu and funnel icon



2. Check the boxes next to the elements that you want to see in the **Probe Manager** list. You can quickly select all elements by checking the **(Select All)** box.
3. Click **OK.** Only the probes corresponding to the elements that you selected now appear in the list. The others have been filtered out.

Note *From the selection menu, you can also decide to sort the column content in alphabetical order or in reverse alphabetical order.*

Adding Probes

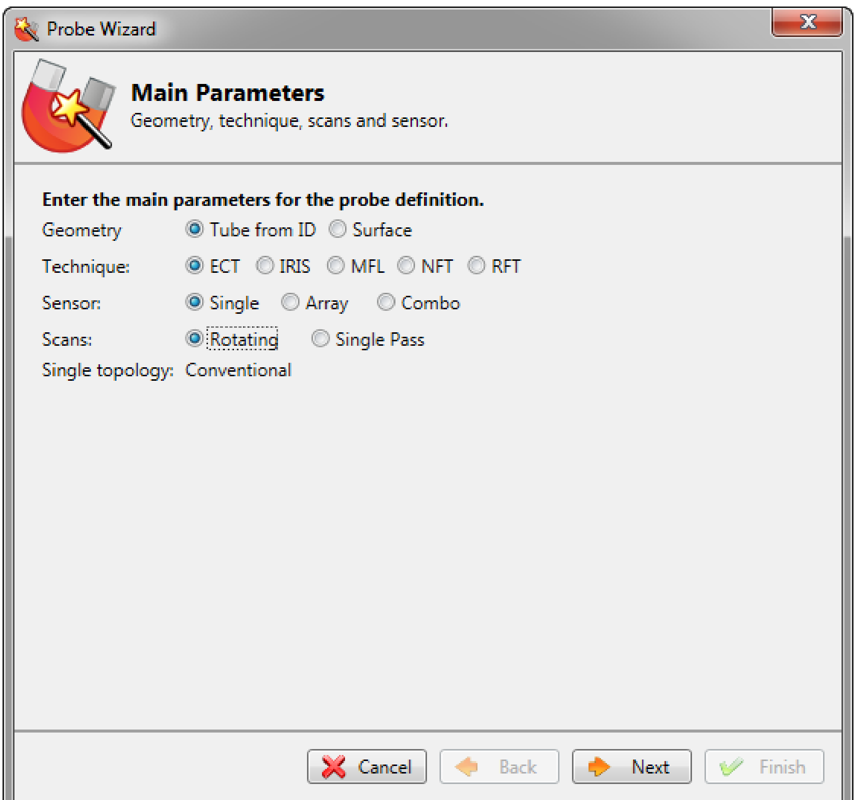
From the **Probe Manager,** you can add a “user” probe via the **Probe Wizard.**

Important The choices offered by the wizard change based on 1) selections made from top to bottom in every “step” and 2) from the previous steps. What is described in the following pages applies to a standard ECT tube inspection probe. Your options may vary.

To do so:

1. In the **Probe Manager** dialog box, click **New**. The **Probe Wizard** dialog box opens.

Figure 3-138 Probe wizard



2. From the first step, select the user probe’s geometry, inspection technique, sensor type, and the scan types to be performed. Click **Next**.

3. For the second step, enter the user probe general information (model name, catalog number [if any], connectors with which it will work, probe diameter, central frequency and, if necessary, a specific description of the probe. Click **Next**.

Note *You can add a new probe model by clicking **New** from that step, or modify the existing model by clicking **Manage**.*

4. For the third step, enter the user probe topology information (use of a pre-amplifier, and the enabling of drivers.) Click **Next**.
5. For the fourth step, enter the user probe channel group information (which are enabled and act as triggers). You can also modify the channels' prefixes and name. Click **Finish**.
   The user probe is added in the **Probe Manager** window and you can now select it for use in the future.

Note *If you loaded a setup containing information about a proprietary probe, it is automatically added to the **Probe Manager**'s list as a user probe.*

Removing Probes

Note *Official probes available in the Eddyfi database cannot be removed; the red X is inactive.*

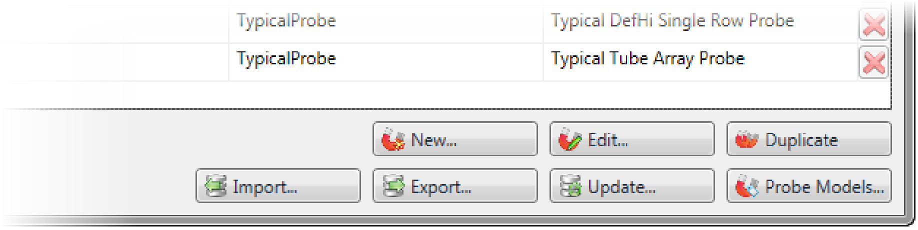
To remove a user probe, simply click the red X at the extreme right of the list in the **Probe Manager.**

Modifying Probes

To modify a user probe:

1. Select the user probe from the **Probe Manager**'s list.
2. Click **Edit**. The **Probe Wizard** appears, where you can make modifications to the selected probe parameters.
3. When you are done making modifications, click **Finish**. The modifications are saved.

Figure 3-139 Probe Manager action buttons



Duplicating Probes

Since Magnifi does not allow you to modify an existing, official Eddyfi probe, if you want to modify one of these probes, you have to duplicate it first.

To do so:

1. Select an existing probe (official or user).
2. Click **Duplicate.** The **Probe Wizard** starts and you can immediately modify parameters.
3. Click **Finish** when you are done. The duplicated probe appears in the list with all appropriate modifications already applied.

Note *This modified probe can be deleted and modified as necessary, contrary to official Eddyfi probes.*

Updating the Probe Database

Eddyfi may occasionally update its official probe database. We recommend that you update the database when such updates are available.

To do so:

1. From the **Probe Manager,** click **Update.** The **Select a New Local Probe Database** window opens.
2. Browse, locate, and select the *.xml* file sent to you by Eddyfi.
3. Click **Open.** The *.xml* file is opened, immediately replacing the official probe database in the process.

Note *Changes made to probe definitions are carried over during probe database updates. You can always return to a probe's original definition when you want to discard your changes.*

Exporting Probe Database

Magnifi allows you to export your database of user probes (for use on another computer, for example). There is no need to export the official database since it is already present with every license of Magnifi.

To export your database of user probes:

1. From the **Probe Manager** window, click **Export.** The **Select a User Probe Database Export Location** window opens.
2. Browse to the location where you want to save the export file (in *.xml* format).
3. Click **Save.** The user probe database is saved in the selected location.

Importing Probe Database

Magnifi allows you to import a database of user probes (coming from another computer, for example). There is no need to import the official database since it is already present with every license of Magnifi. The official database is only updated, as explained previously (see page 194).

To import a user probe database:

1. From the **Probe Manager** window, click **Import.** The **Select a User Probe Database to Import** window opens.
2. Browse to the location where you the file (in *.xml* format) to import is located.
3. Click **Open.** The user probe database opens and the probe database is modified accordingly.

Managing Probe Models

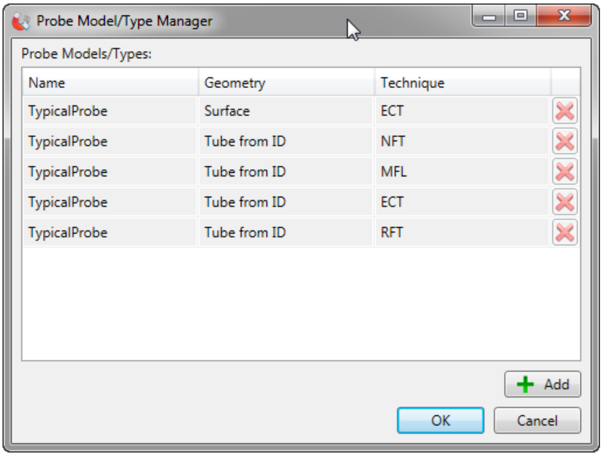
With the Probe Manager, you can not only manage individual probes, but also probe models.

To do so:

1. From the **Settings** menu, select **Probe Database.** The **Probe Manager** window appears. In this window, you see all default Eddyfi probes.
2. Click **Models/Types.** in the lower-right corner. The **Probe Model/Type Manager** window appears, where you can manage user probe models.

   Probe models/types with a grey background are standard Eddyfi probes. They cannot be modified or deleted.

Figure 3-140 The **Probe Model/Type Manager** window



3. Click **Add.** A new line appears at the end of the list with the name **New Probe Model** highlighted.
4. Enter a name for the new probe model and press the TAB key. The **Geometry** column is highlighted.
5. Select whether the new probe model is for tube (**Tube from ID**) or surface (**Surface**) inspection and press the TAB key.
6. If you selected **Surface,** the **Technique** column turns grey because the only technique supported right now is ECT. If you selected **Tube from ID,** you can select one of four techniques (ECT, RFT, NFT, MFL).
7. Once you are done defining your probe model, click **OK.** The window closes, and the new model will be available in the **Probe Wizard.**

Managing Indication Codes

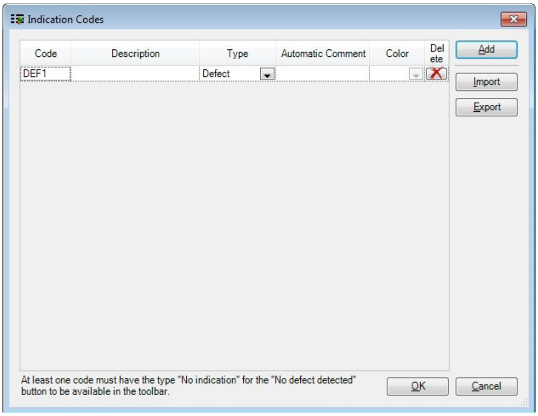
Indication codes appear in the **Code** pane as well as in C-scans. They are generally abbreviations destined to highlight specific indications found during an inspection.

Adding Indication Codes

To add indication codes:

1. On the **Settings** menu, click **Indication Codes.** The **Indications Codes** window appears. By default, indication code **DF1** and **DF2** already exist.

Figure 3-141 The **Indications Codes** window



2. Click **Add.** By default, a **CODE1** code appears. It is editable.
3. Enter the code that you want to replace **CODE1.**
4. Enter a description for that code in the **Description** column.
5. Select the type of indication in the **Type** list.
   - **Defect:** indication that is not man made and is usually the result of wear and tear on the part (pitting, corrosion, etc.). Such code types, when added to the report, will include signal measurements and sizing (if sizing curves are defined.)

- **No Indication:** signifies that no indications were found in that inspected tube or surface. You must create one of this type of indication to have access to the green check mark in the Indication Codes toolbar (for more information, see "From the Indication Codes Toolbar" on page 278.

Note *Only one code can be associated with the No Indication type.*

- **Feature:** an indication that describes a characteristic of the tube, but for which there is no measurements required (plugged, obstructed, suspicious, dirty, rough, etc.). Selecting a feature also activates the **Color** flag column (see below).

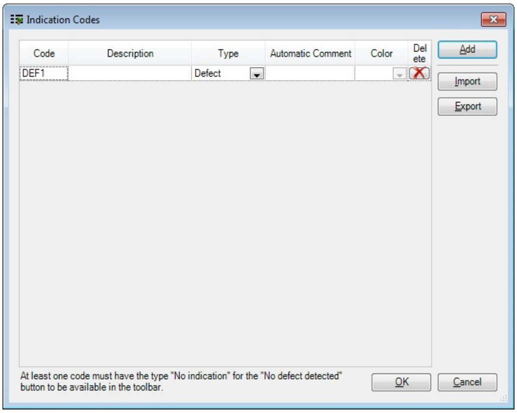
6. If you want to add text automatically in the Comments column whenever you add this code to a report, enter it in the **Automatic comment** column.
7. When you select **Feature** in the **Type** column, the **Color** flag column becomes active. This assigns a color to the icon that will be added to the Indication Codes toolbar. Each feature code has its own icon in this toolbar.
8. Repeat steps 2 and 7 until you have added all the indication codes that you will need.
9. Once you are done adding indication codes, click **OK.** The codes in the list will be available when identifying indications during analysis.

Importing Indication Codes

To import indication codes:

1. On the **Settings** menu, click **Indication Codes.** The **Indications Codes** window appears.

Figure 3-142 The **Indications Codes** window



2. Click **Import.** A standard **Open** dialog box appears, where you can browse to locate the file (in *.xml* format) containing the indication codes.
3. Click **Open.** The codes are loaded in the **Indication Codes** window.

Removing Indication Codes

To remove indications codes:

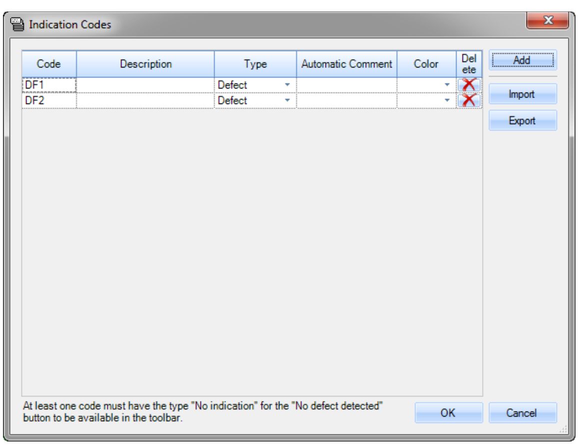
1. From the **Settings** menu, select **Indication Codes.** The **Indication Codes** window appears.
2. Select the indication code that you want to remove and click the red X in the **Delete** column. The selected indication code disappears from the list.
3. Repeat step 2 until you have removed all the indication codes that you want to remove.
4. Once you are done removing indication codes, click **OK.**

Exporting Indication Codes

To export indication codes:

1. From the **Settings** menu, select **Indication Codes.** The **Indications Codes** window appears.

Figure 3-143 The **Indications Codes** window



2. Click **Export.** A standard **Save** dialog box appears, where you can browse to the location where you want to save the indication codes file (in *.xml* format).
3. Click **Save.** The indication codes are saved at the selected location.

Setting Up Sizing Curves

Sizing curves are used to correlate signal parameters to a physical dimension. For example, in tube inspection, signal phase and/or amplitude can be used to determine the depth of an indication (or the percentage of wall loss).

In Magnifi, a sizing curve is created in four simple steps:

1. Configuration of general sizing curve parameters
2. Configuration of calibration points
3. Configuration of individual sizing curves
4. Calibration

Configuring General Sizing Curve Parameters

To configure sizing curves:

1. Select **Sizing Curves** from the **Setup** tab in the Control bar, or from the **Settings** menu. The **Sizing Curves Configuration** window appears.

Figure 3-144 Selecting **Sizing Curves**

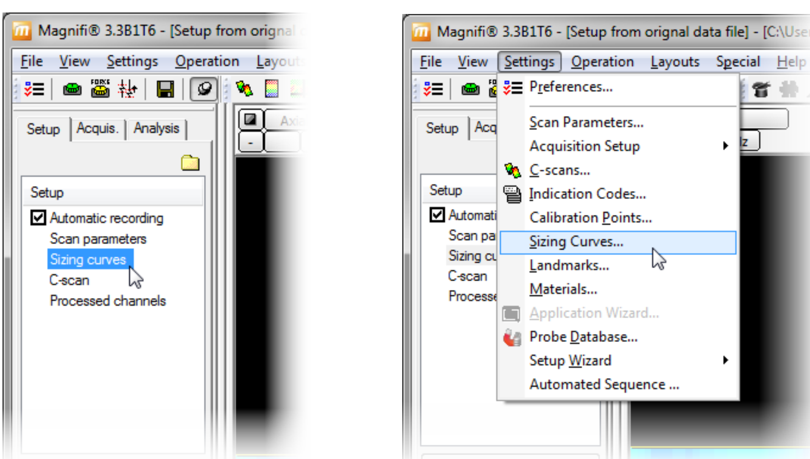
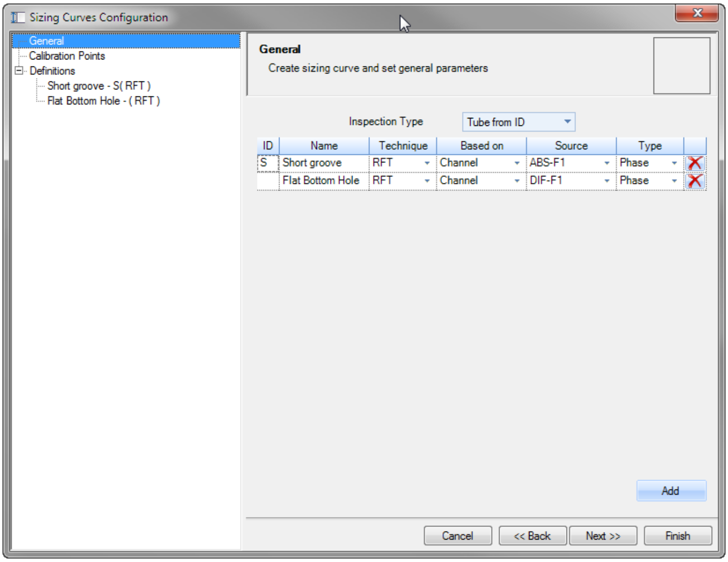


Figure 3-145 The **Sizing Curves Configuration** window



2. From the **General** page, select the type of inspection that you want to perform from the **Inspection Type** drop-down list:
   - **Plate**
   - **Tube from ID**
   - **Tube from OD**

Note *The type of inspection selected will be applied to all sizing curves defined in the setup.*

3. Click **Add** to add the sizing curve to the list.
4. Configure the added sizing curve:
   - **a** Enter a one- or two-character identification in the **ID** field (optional, but very useful if you have more than one curve for the same channel or C-scan).
   - **b** Enter the name of the sizing curve in the **Name** column.
   - **c** Select the inspection technique in the **Technique** drop-down list.
   - **d** Select what the sizing curve will be based on from the **Based on** drop-down list. Sizing can be based on channels or C-scans along the X or Y axis.
   - **e** Select the source of the sizing curve data in the **Source** drop-down list.
   - **f** Select the type of sizing curve (Amplitude or Phase) in the **Type** drop-down list.
5. Repeat steps 2 to 4 to add as many sizing curves as needed and, when done, click **Next>>**.

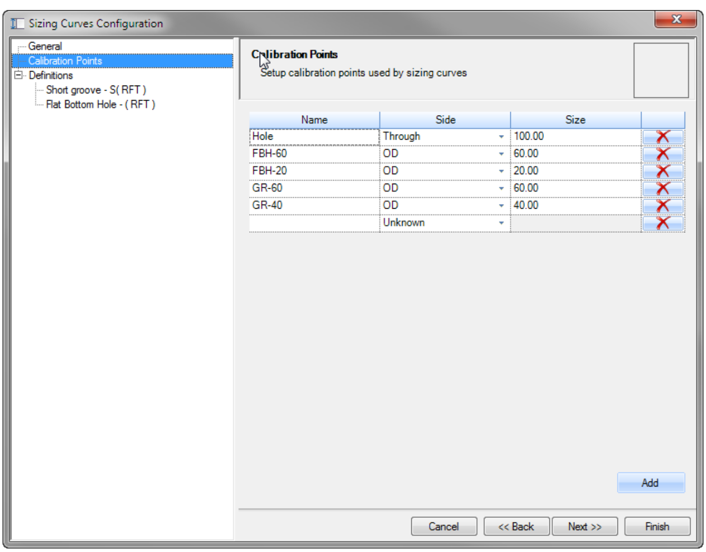
Configuring Calibration Points for Sizing Curves

To configure calibration points:

1. In the **Calibration Points** page, add all calibration points used in your calibration sample and fill the name, side, and size of these calibration points. In the **Side** column, the available choices depend on the inspection type selected in the previous step:
   - **Through:** defect that is going through the entire thickness of the inspected part (applies to plates as well as tubes).
   - **Far:** defect located on the far side of the inspected part (appears as **OD** for tubes).
   - **Near:** defect located on the near side of the inspected part (surface breaking defect) (appears as **ID** for tubes).
   - **None:** when the technique used cannot discriminate the location of the defect (like with RFT) (applies to plates as well as tubes).
   - **Unknown:** For reference. Not used for sizing. (applies to plates as well as tubes)

Note *In the **Size** fields, the values entered are unit-less. The work unit is defined in the next step.*

Figure 3-146 The **Calibration Points** page from the **Sizing Curves Configuration** window



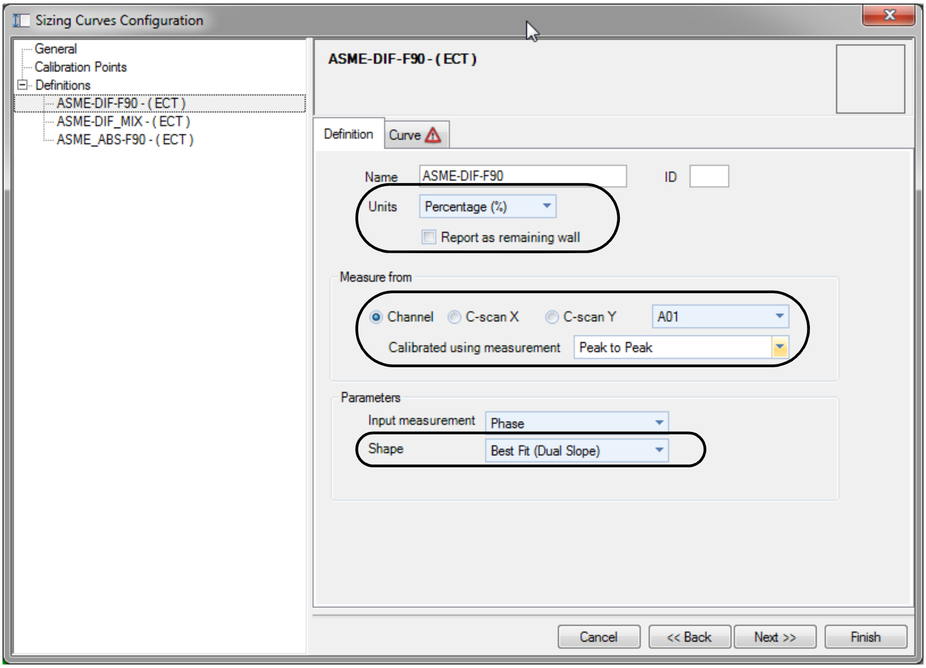
2. Once all your calibration points are defined, click **Next>>**.

Configuring Individual Sizing Curves

Complete the channel definition for each calibration point. The amount of information to configure depends on the inspection technology used.

The **Name, ID** and **Input measurement** fields are already configured from a previous step in the wizard.

Figure 3-147 Channel parameter definitions for calibration points – ECT technology



To configure channel information for your sizing curves:

1. Select the unit of work to use on the **Units** list. The selected unit applies to the **Size** column in the **Calibration Points** step of the wizard. The **Units** list allows you to select an absolute (e.g., **mm**) or relative (e.g., **Percentage (%)**) unit of measurement. This gives you the amount of matter removed from the tube.

Note *If you want to know the amount of remaining matter, select the **Report as remaining wall** check box.*

2. In the **Measure from** group, select the channel used for measurement.
3. In the **Calibrated using measurement** list, select the appropriate calibration measurement technique (varies according to the input measurement; phase or amplitude):
   - **Absolute**
   - **Absolute Horizontal**
   - **Absolute Peak**
   - **Absolute Peak Horizontal**
   - **Absolute Peak Vertical**
   - **Absolute Vertical**
   - **Horizontal**
   - **Maximum Rate**
   - **Peak to Peak**
   - **Peak to Peak First Transition**
   - **Vertical**

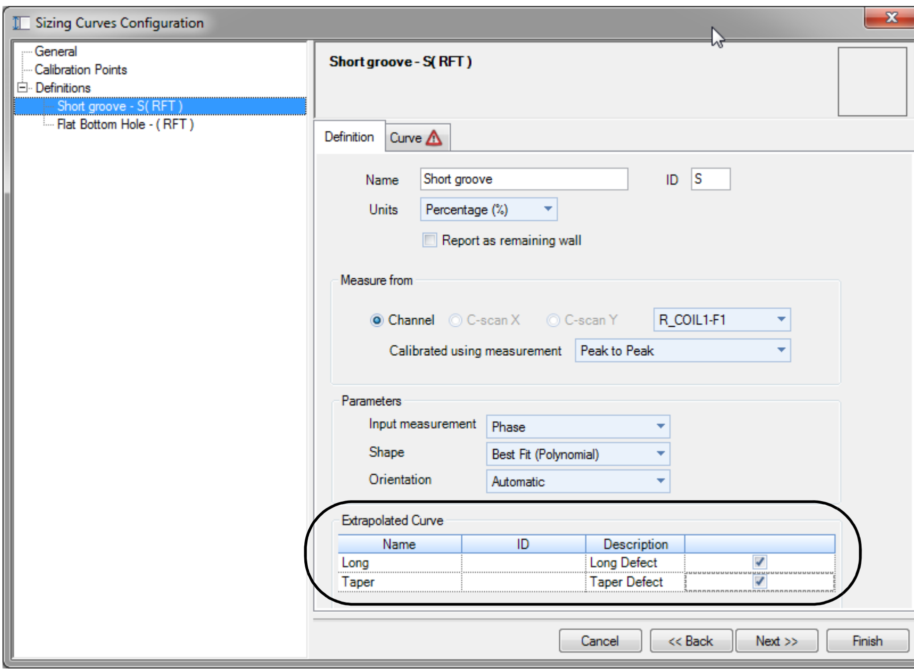
Note *For absolute channels, we recommend that you use **Peak to Peak First Transition** instead of **Peak to Peak**.*

4. In the **Parameters** subsection, select the shape of the sizing curve that you want to create (the **Input measurement** is already selected from a previous step in the wizard):
   - **Best Fit (Dual linear)** (*for phase measurement only*): an ECT curve with two linear segments representing ID and OD (or Near and Far) side calibration points in relationship with phase.
   - **Best Fit (Dual Slope)** (*for phase measurement only*): an ECT curve with two segments representing ID and OD (or Near and Far) side calibration points' relationship with phase. The OD side of the curve will need at least three points (including the hole) in order to trace a polynomial curve.
   - **Best Fit (Linear)** (*for phase and amplitude measurements*): best linear interpolation within the measured calibration points.

- **Best Fit (Polynomial)** (*for phase and amplitude measurements*): best polynomial (degree 2) interpolation within the measured (at least three) calibration points.
- **Connected Points** (*for phase and amplitude measurements*): simple, point-to-point curve.

5. If you are working with RFT technology and absolute channels in combination with voltage planes, there is one more section to configure: **Extrapolated Curve.** In this section, you give a name, ID, and description to your RFT extrapolated curves, and select whether to use them or not.
   The extrapolation can be used to evaluate the depth of long and taper defects from a curve made of short defects.

Figure 3-148 Channel parameter definitions for calibration points – RFT technology



6. Once you are done, click the **Curve** tab. The **Curve** page appears.

Note *If you selected an invalid curve shape in step 4, or if the curve has not been calibrated yet, the word **INVALID** appears instead of a curve on the **Curve** page.*

Figure 3-149 Channel curve definition for calibration points – ECT technology (dual slope curve)

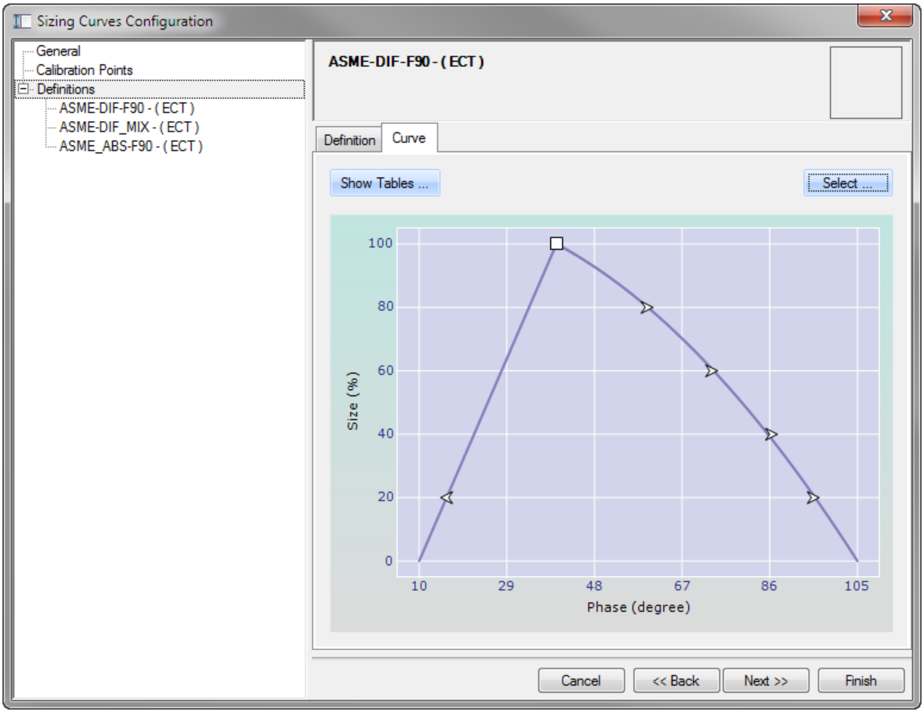
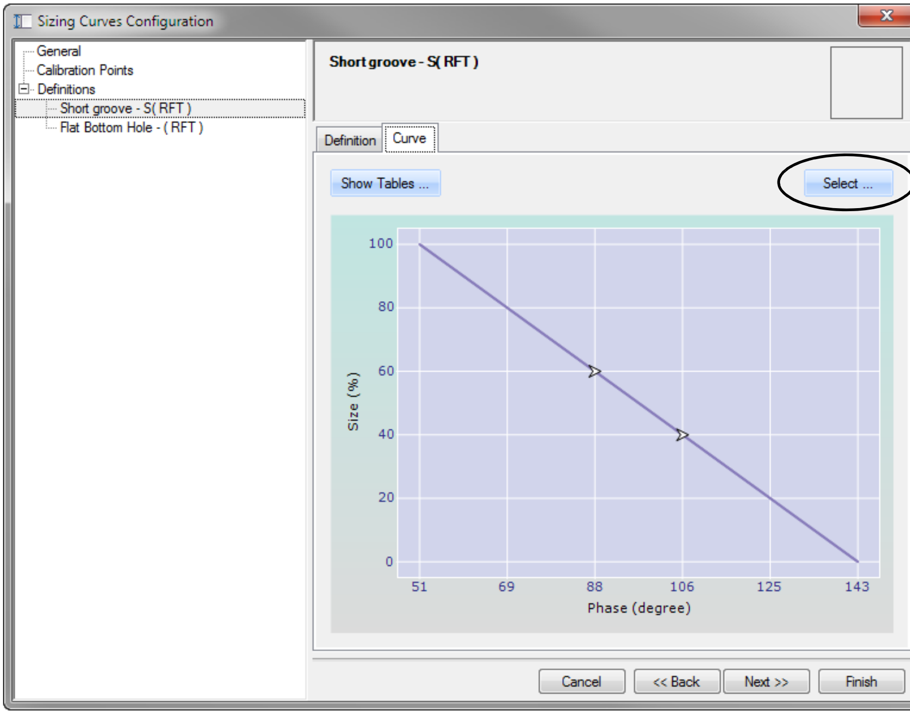
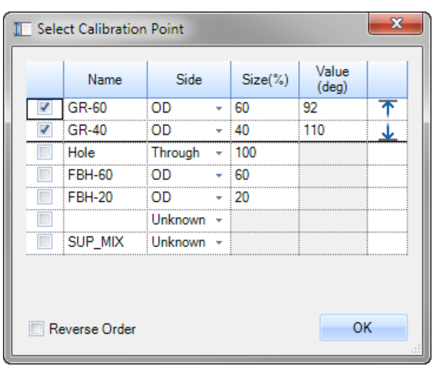


Figure 3-150 Channel curve definition for calibration points – RFT technology (linear slope)



7. Click **Select.** The **Select Calibration Point** window appears.

Figure 3-151 The **Select Calibration Point** window



8. Select all calibration points required to create a sizing curve and click **OK.** Selected points will be grouped and sorted at the top of the window. The sorting depends on the side of the indication. To have a valid curve, the sorting should provide an increasing phase or amplitude value once the calibration has been performed. If the phase or amplitude value measured during calibration is decreasing, the curve will be invalid. You can change the sorting by checking the **Reverse order** box.
9. Click **Finish.** Your sizing curve is defined.

Calibrating the System and Verifying Curves

Once you have configured your curves, you have to calibrate your system and verify the resulting curves.

For more information on calibrating the system, see "Calibrating Systems" on page 238.

Once your system has been calibrated:

1. Select **Sizing curves** from the Control Bar. The **Sizing Curves Configuration** window appears.
2. Under **Definitions,** select the curve name that you want to review, and click the **Curve** tab. A valid curve should appear.

Note *If the word **INVALID** still appears, at least one calibration point was not measured properly during the calibration, or the order of the indication codes is incorrect. Click **Select** and manually correct the invalid calibration points in the **Select Calibration Point** window that appears (see Figure 3-151).*

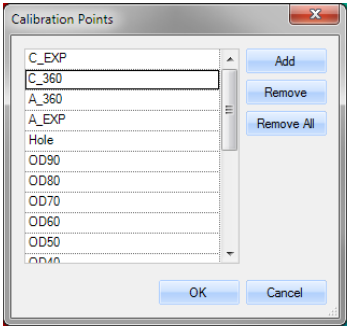
Managing Calibration Points

Calibration points are predetermined reference points used to calibrate channels and C-scans, and to build sizing curves. They are usually managed when setting up sizing curves (see "Setting Up Sizing Curves" on page 200), but they can also be added in the **Calibration Points** window.

To add calibration points:

1. From the **Settings** menu, select **Calibration points.** The **Calibration Points** window appears.

Figure 3-152 The **Calibration Points** window



2. Click **Add.** The first available field in the left column becomes editable.
3. Enter a name for this calibration point.
4. Repeat steps 2 and 3 until you have added all the calibration points that you will need.
5. Once you are done adding calibration points, click **OK.** The calibration points in the list will be available everywhere reference points are needed.

To remove calibration points:

Note *It is not possible to remove a calibration point that is already used for a channel or C-scan calibration or for a sizing curve.*

1. From the **Settings** menu, select **Calibration points.** The **Calibration Points** window appears.
2. Select the calibration point that you want to remove and click **Remove.** The calibration point selected disappears from the list.
3. Repeat step 2 until you have removed all the indication codes that you want to remove.

Note *To quickly remove all calibration points, click **Remove All.***

4. Once you are done removing indication codes, click **OK.**

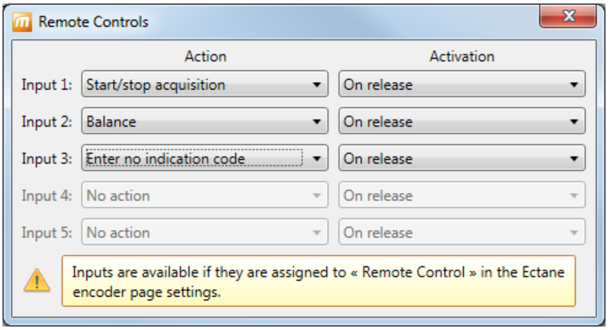
Managing Remote Controls

To access remote controls in Magnifi, on the **Settings** menu, click **Remote controls**.

Inputs are only available when the associated digital input (configured on the **Encoder** menu) is set to **Remote control**.

In the example below, it is possible to program the first three inputs with one of the functions presented here:

Figure 3-153 Remote Controls dialog box



- **No action**: The input is available, but has no function.
- **Balance**: Balances the probe
- **Clear screen**: Clears the Lissajous (only during acquisition)
- **Start acquisition**: Starts data acquisition. When the automatic recording mode is enabled, the function also starts recording data to a file.
- **Stop acquisition**: Stops data acquisition. When the automatic recording mode is enabled, the function also stops recording data to the file. **Stop acquisition** is only available after **Start acquisition**.
- **Start/stop acquisition**: Toggles acquisition on or off each time you activate the remote control. If automatic recording is enabled, it also record data to a file.
- **Start recording**: Starts recording data to a file. The function is only available after starting acquisition when the automatic recording function is disabled.
- **Stop recording**: Stops recording data to a file. This function is only available after starting recording when the automatic recording function is disabled.
- **Start/stop recording**: Toggles recording on or off each time you activate the remote control. The function is only available after starting acquisition when the automatic recording function is disabled.
- **Start next file**: Starts the next file in the inspection list. If automatic recording is enabled, it also starts recording data to the file.

- **Select previous file**: Selects the file before the one currently selected in the inspection list.
- **Select next file**: Selects the file next to the one currently selected in the inspection list.
- **Enter no indication code**: Adds a *No indication* code to the report and automatically start the next file.

For each selected action, you must specify when to perform it — **On press** or **On release** of the remote control.

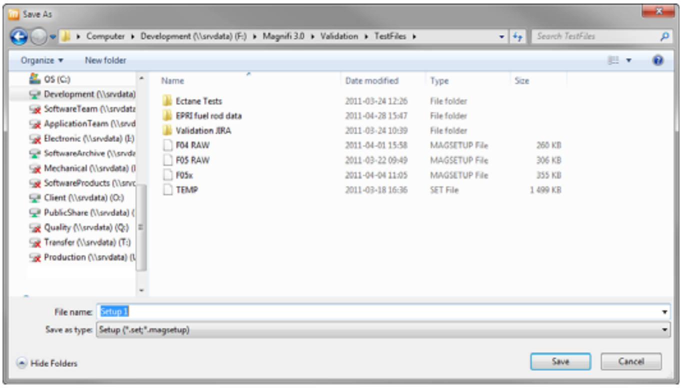
Saving Setups

An acquisition setup is comprised of all the numerical parameters set in the previous pages (scan, acquisition, and C-scan parameters, sizing curves, processed channels, calibration points, etc.) After you have finished configuring an acquisition setup, you should save it for future use.

To save a setup:

1. On the **File** menu, click **Save Setup**. A **Save As** dialog box appears.

Figure 3-154 Saving a setup



2. Browse to the directory where you want to save your setup file.
3. Enter the name that you want to give to the setup in the **File name** field and click **Save.** The setup file is saved in the directory that you chose, under the name that you entered.

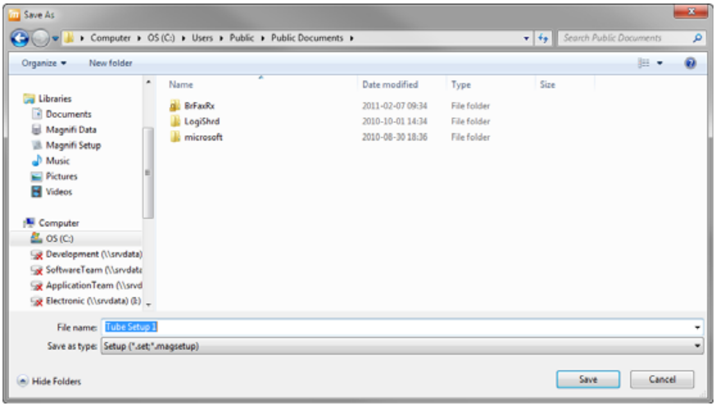
Saving a Setup under a Different Name

If you need to create a setup and would like to base it on an existing setup, you can simply load an existing setup and save it under a different name. From there, you can modify only the parameters that need to be modified in the newly saved setup.

To do so:

1. With a setup loaded (that you either loaded specifically for this operation, or that you already have on screen), select **File>Save Setup As.** A standard **Save As** dialog box appears.

Figure 3-155 The **Save As** dialog box



2. In the dialog box, browse to the location where you want to save the setup.
3. Enter the name of the new setup in the **File name** text box.
4. Click **Save.** The new setup file (*.*magsetup*) is saved at the location that you chose.

Creating Inspection Projects

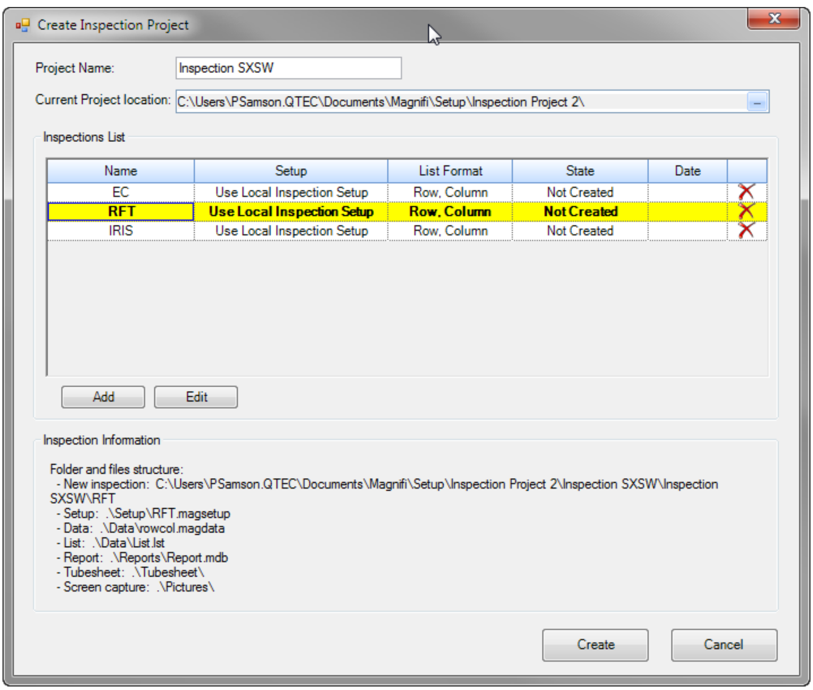
Before you start a new inspection project, you have to determine what will happen to the files created during that project. Inspection projects can be varied (client, outage, inspection type, etc.), and there constituting inspections, just as such.

You create inspection projects through the **Create Inspection Project** window.

To do so:

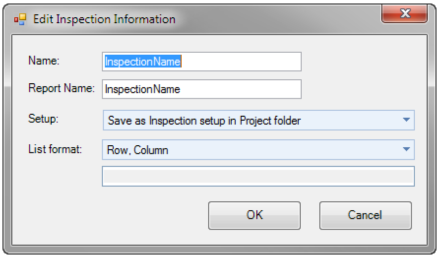
1. From the **File** menu, select **Create Inspection Project.** The **Create Inspection Project** window appears.

Figure 3-156 The **Create Inspection Project** window



2. Enter the name of the inspection project in the **Project Name** text box.
3. Click the Browse button next to the **Current Project location** text box. A standard dialog box appears, where you can browse to the location where you want to store your project files.
4. Click **Add** to add an inspection in the **Inspections List.** The **Edit Inspection Information** window appears.

Figure 3-157 The **Edit Inspection Information** window



5. Enter the inspection name in the **Name** text box.
6. Enter the report name in the **Report Name** text box.
7. In the **Setup** drop-down list, select how you want to manage the inspection setup file:
   - **Save as Inspection Setup in Project Folder:** (default value) copy and rename the current setup and saves it under the project setup directory.

Important When you use this option, you must already have the appropriate setup loaded in Magnifi.

   - **Keep current name and location:** performs no action on the setup file. The current name is kept and the file is not copied in the project setup directory. This is the option to use if you prepare in advance several inspections and the current Magnifi setup is not necessarily the one to be used for this inspection.
8. In the **List format** drop-down list, select the list format that you want to use:
   - **Row, Column**
   - **Zone, Row, Column**
   - **Free format**
   - **Import**

     This last option allows you to select a **.lst* file, using the **Browse** button. This file will be copied in the Data folder of the inspection project directory, and renamed *List.lst*. Its name and location will be displayed in the text box underneath the **List format** drop-down list.
9. Repeat steps 4 to 8 until all your inspections for this inspection project are configured.
10. Once you have set the inspections, information about the folder and file structure appears in the **Inspection Information** subsection. The **New inspection** line indicates the directory where the new inspection files will be saved, and the lines underneath, the specific location of each file within the inspection directory. If this is to your liking, click **Create.** The folder and file structure is created. The highlighted inspection is automatically loaded in Magnifi, and you are ready to start inspections.

Performing Acquisitions

4

Setting up the Ectane

When you are ready to perform acquisitions, the first step is to prepare the Ectane test instrument as explained in Chapter 2 of the Ectane instruction manual.

Once the Ectane is properly positioned, and physical connections are established between the instrument and the workstation, you need to establish the software connection.

Establishing Communication with Ectane Instruments

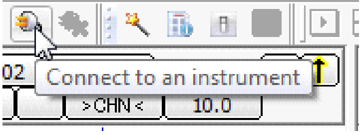
When using an Ectane instrument, Eddyfi recommends that your network card be configured to obtain an IP address automatically. This will ensure a truly plug-and-play experience with the instrument.

Note *You can also use your workstation with older instruments (e.g., MS5800™ and TC7700™). For more information on that topic, see* *"Establishing Communication with Magnifi" on page 313.*

To establish communication with the Ectane:

1. Start the Ectane as explained in the Instruction Manual.
2. Start the Magnifi software.
3. Click **Connect to an instrument**.

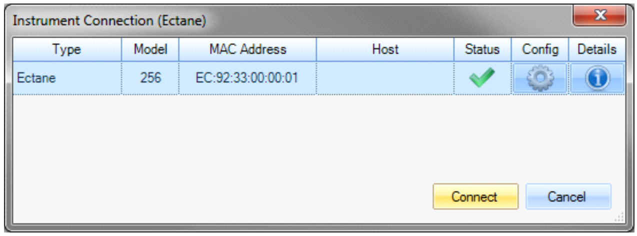
Figure 4-1 Connecting to an Instrument



Note *If the* ***Connect to an instrument*** *button is not active, select* ***New>Ectane Setup*** *from the* ***File*** *menu.*

4. Select the available instrument, and click **Connect** (see Figure 4-2).

Figure 4-2 Selecting an instrument

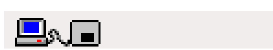


Note *If no Ectane is present in the* ***Instrument Connection*** *window, or if the* ***Status*** *box contains a red X, see “Quick Troubleshooting Guide” on page 288. A green check mark or a yellow exclamation mark are both valid communication statuses.*

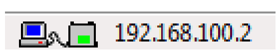
The communication status between Magnifi and the Ectane is known from the icons located in the status bar, at the lower right corner of the screen.

A temperature indicator is displayed when the connection is established. This represents the internal temperature of the instrument. (Does not work with third-party equipment.)

No instrument is available.

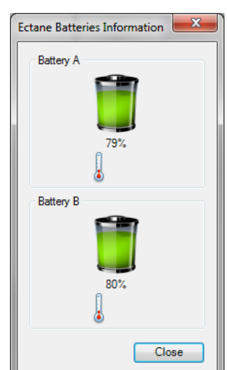


An instrument is active and ready for connection



An instrument is connected and ready for acquisition

A battery indicator is seen when the connection is established. Double-clicking the battery provides more details about the battery status:



Loading Setup Files

After establishing communications with your instrument, it is now time to start working with a setup file.

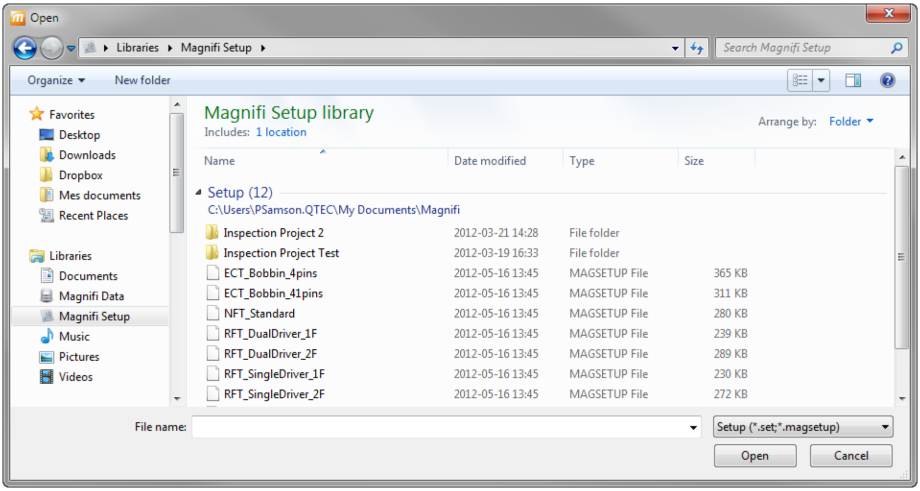
You can:

- create a new setup (see “Configuring Ectane ECT Acquisition Setups” on page 81);
- load an existing setup; or
- load a recently used setup file.

Loading Existing Setup Files

To load an existing setup, select **Load Setup** from the **File** menu. A window opens, where you can select from many existing setups. You can also browse through directories to find the setup file that you want.

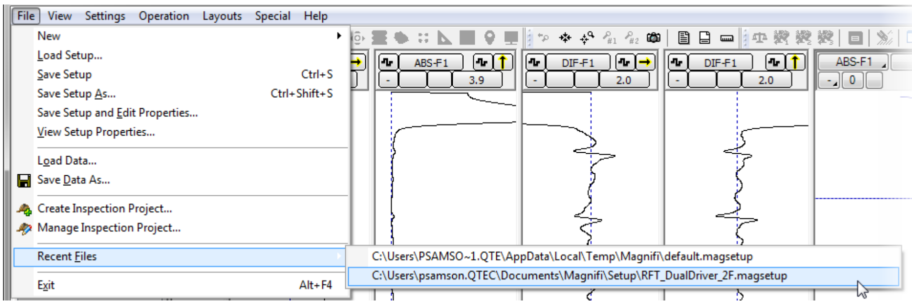
Figure 4-3 The **Open** window



Loading Recently Used Setup Files

If you recently used a setup file, you can quickly access it by selecting it from the **Recent Files** item in the **File** menu.

Figure 4-4 Selecting a recent file



Converting Legacy Acquisition Setup Files to Ectane Files

To use a TC7700 or MS5800 acquisition setup with an Ectane instrument, you have to convert the setup file first.

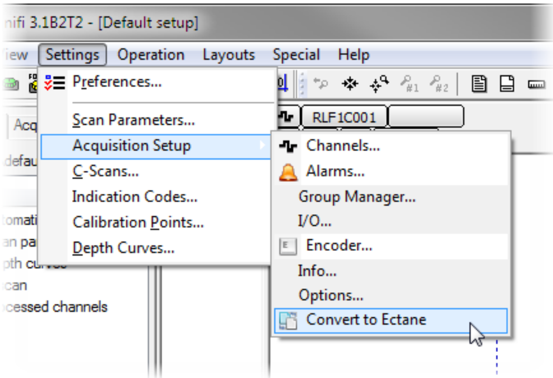
Note *Only parts of the original TC7700 and MS5800 setup files are converted. Features not supported by the Ectane instrument (alarms, group managers, data compression, report previews, and probe state) are not taken into account. At the time of writing, the only parameters kept are the channel, encoder, and I/O data.*

To do so:

1. Select **Load Setup** from the **File** menu. A standard **Open** dialog box appears.
2. Browse through this dialog box until you find the acquisition setup file (*.*set*) that you want.
3. Select the file and click **Open.** The setup file loads in Magnifi.

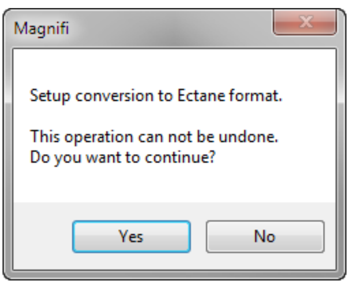
4. From the **Settings** menu, select **Acquisition Setup>Convert to Ectane.**

Figure 4-5 Selecting **Convert to Ectane**



5. In the **Magnifi** dialog box that appears, click **Yes** to continue.

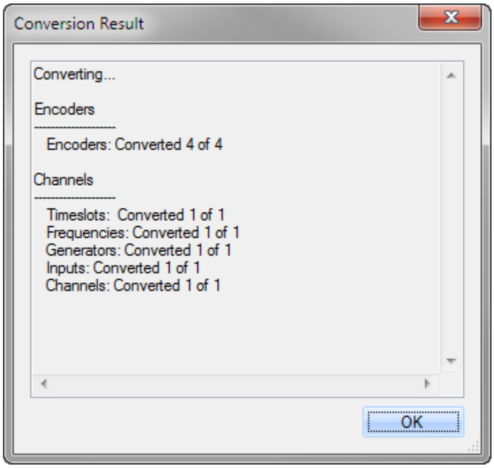
Figure 4-6 The **Magnifi** confirmation dialog box



6. Once the conversion is complete, a **Conversion Result** dialog box appears.

Important Read the content of the **Conversion Result** dialog carefully. All incompatibilities and non-converted items will be listed and briefly explained.

Figure 4-7 The **Conversion Result** dialog box



7. Click **OK** to dismiss the dialog box and complete the conversion.

Important If you want to keep the original acquisition setup file after conversion, you *must* use **Save Setup As** from the **File** menu. Otherwise, if you simply select **Save Setup** from the **File** menu, the original TC7700 setup file will be overwritten *permanently*.

Fine-Tuning Converted Setups

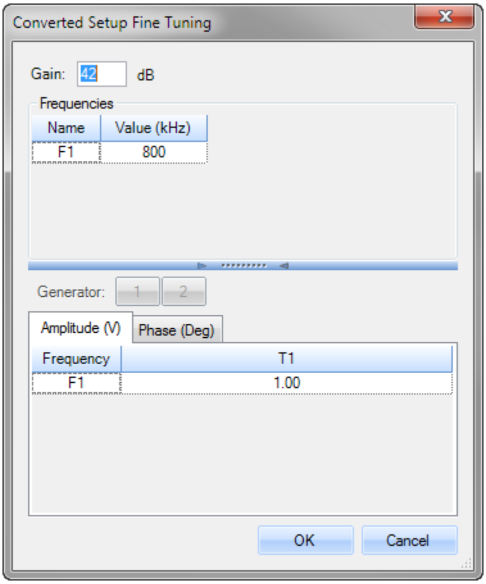
After you have converted an existing setup file, there is a number of parameters that you cannot change. However, certain basic parameters can still be fine-tuned:

- Gain
- Frequencies
- Amplitude
- Phase

To fine-tune these parameters:

1. From the **Settings** menu, select **Acquisition Setup>Converted Setup Fine Tuning.** You can also click the **Converted setup fine tuning** button in the Acquisition Setup toolbar. The **Converted Setup Fine Tuning** window appears.

Figure 4-8 The **Converted Setup Fine Tuning** window



2. To change the gain, double-click in the **Gain** field and enter a new value.
3. To change the frequencies, double-click the values to change in the **Value (kHz)** column and enter new values as required.
4. If you use more than one generator, click the appropriate generator number adjacent to the **Generator** label.
5. To change the amplitude, select the **Amplitude (V)** tab and double-click the values to change in the **T*n*** column.
6. Enter new values.
7. To change the phase, select the **Phase (Deg)** tab and double-click the values to change in the **T*n*** column.
8. Enter new values and click **OK.**

Performing Acquisitions in Setup Mode

You can perform basic acquisitions while still in Setup mode. This can be useful when you need to see what acquisition data will look like, but in a more information-oriented purpose, like when you are fine-tuning a setup.

Performing acquisitions in Setup mode allows you to perform a system calibration and change acquisition parameters, some of which you cannot change once you are in Acquisition mode. Data collected in Setup mode is not saved. For more information on saving data, see “Saving Data” on page 228.

Before you start an acquisition, you have to balance your inspection hardware.

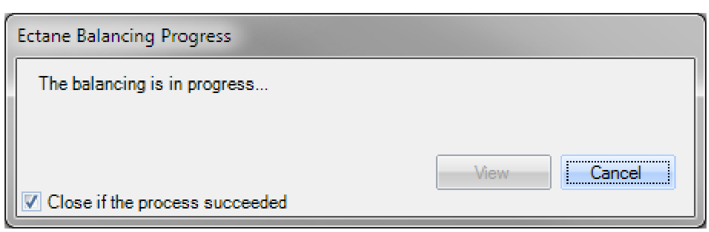
Balancing

The purpose of balancing is to eliminate the signal seen by the probe when it is placed over a sound area of a part to inspect. This operation resets the signal and allows the system to operate over its maximum dynamic range. This is an operation better performed right before commencing a large number of scans.

To balance your system, place the probe over a sound area and click **Unit balancing** from the Acquisition toolbar, select **Balancing** from the **Operation** menu, or press F6.

A window appears briefly, indicating that balancing is in progress.

Figure 4-9 Balancing in progress

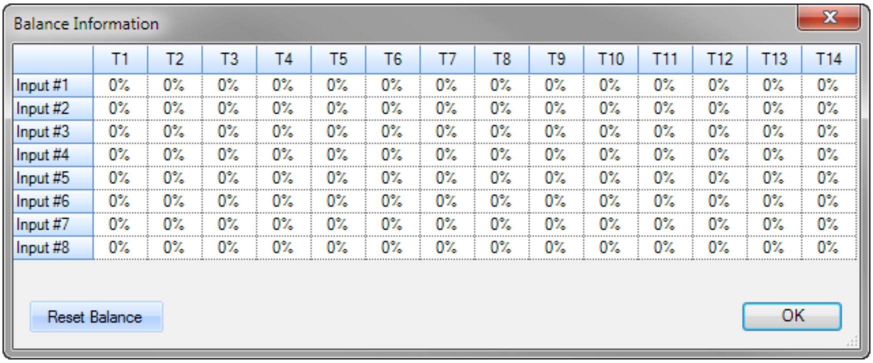


Once the system is balanced, the window disappears and the system is ready for acquisitions. If the balancing operation fails, see “Maintenance & Troubleshooting” on page 287.

Displaying Balancing Information

You can view the balancing information used by the system by selecting **Balance Information** from the **Special** menu. This opens a window indicating balancing values for the various inputs. You can also reset the balance information by clicking **Reset Balance.**

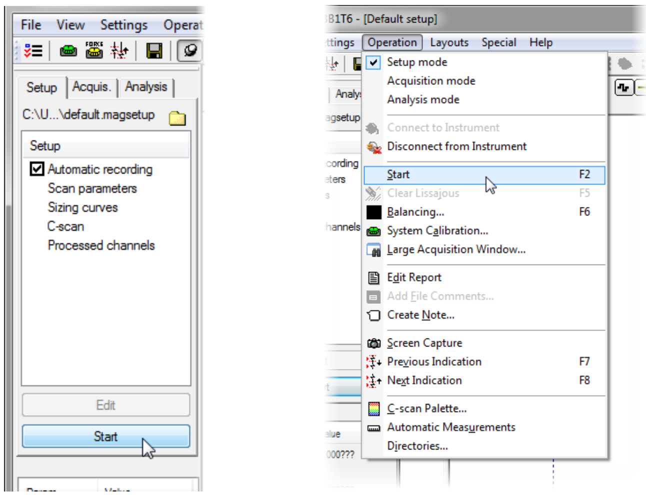
Figure 4-10 The **Balance Information** window



Starting an Acquisition in Setup Mode

To start an acquisition, click **Start** from the Control bar, select **Start** from the **Operation** menu, or press F2.

Figure 4-11 Ways to start an acquisition



Data starts appearing in the different views as you scan.

Once you are done scanning, click **Stop** from the Control bar, select **Stop** from the **Operation** menu, or press F2 again. Data is processed as per your setup, and then displayed.

For more information on data analysis, see “Analyzing & Reporting Data” on page 255.

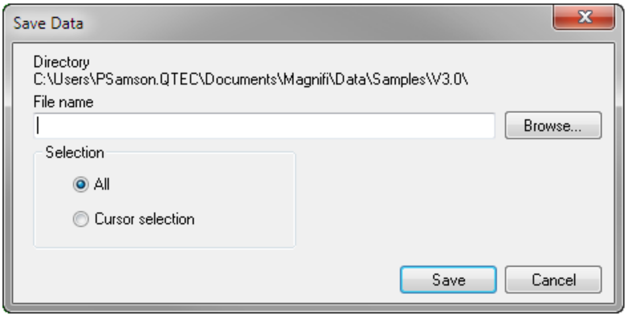
Saving Data

Normally, data are saved in the directories associated with the inspection project in use. You can, however, save data in another directory if you wish.

To do so:

1. Select **Save Data As** from the **File** menu. A **Save Data** window appears.

Figure 4-12 The **Save Data** window



2. Enter a name for the file in the **File name** text box.
3. Click **Browse** to access the directory where you want to save this data file.
4. From the **Selection** subsection, select whether you want to save all data, or just the data found between the cursors.
5. Click **Save.** The data that you selected is saved in the location that you chose.

Performing RFT-Specific Tasks

Before performing an official inspection with the remote-field technique (using the voltage plane), you need to:

- Normalize the signal
- Store the nominal tube response

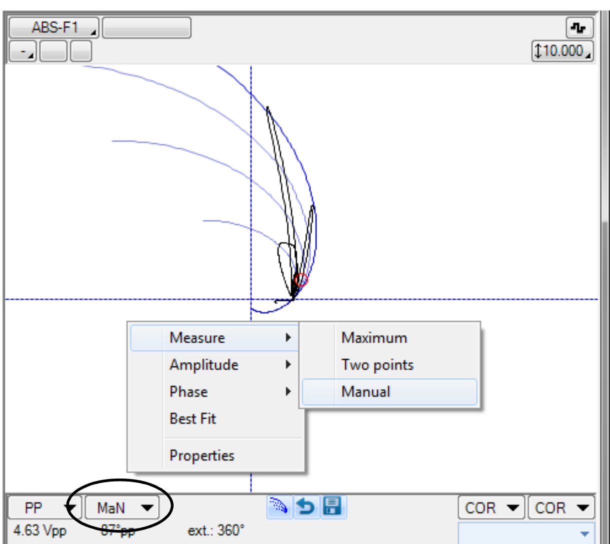
Normalization

Note *For normalization to function properly, a voltage plane must first exist on the **Processed Channel** menu.*

To normalize the signal:

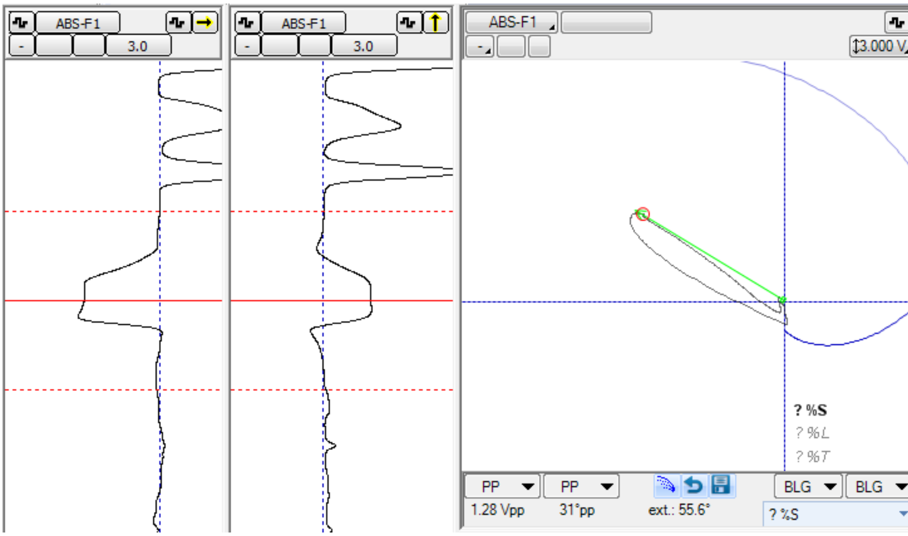
1. Acquire a calibration standard that contains a simulated tube support plate (TSP).
2. With the data selection cursor, select the TSP signal and adjust the scale in the voltage plane view so as to have a good view of the signal.
3. Select the manual measurement mode by right-clicking the voltage plane, pointing to **Measure,** and then clicking **Manual**.
   Alternately, you can click the manual button at the bottom of the view.

Figure 4-13 Selecting manual measurements



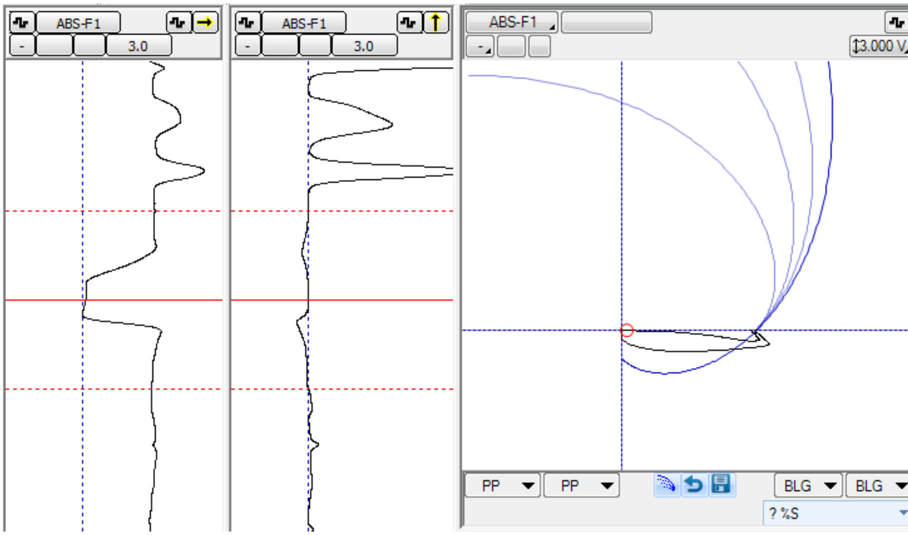
4. Draw a vector from the beginning of the nominal tube signal to the tip of the TSP.

Figure 4-14 Drawing a vector



5. Click the Normalization button at the bottom of the voltage plane view. This adjusts the rotation, gain, and translation along the X and Y axes to bring the nominal tube signal at the bottom of the skin depth spiral, and the tip of the TSP at zero.

Figure 4-15 Results from a normalization



Storing Nominal Tube Response

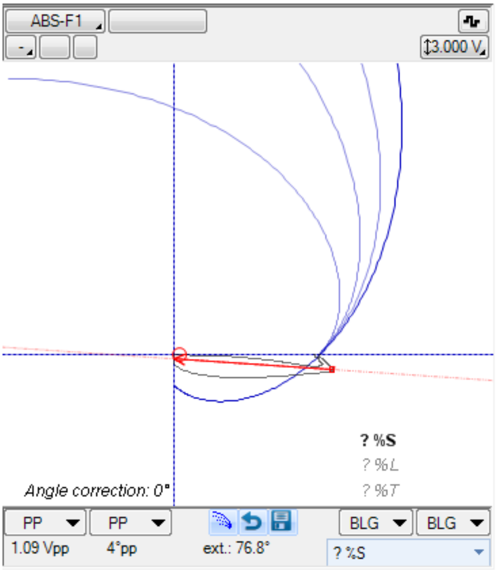
To store the nominal tube response:

1. Perform a normalization (see “Normalization” on page 229).

2. Click the Store nominal button at the bottom of the voltage plane view. The current normalization settings are stored and the tracking of the angle correction begins; it will be performed every time a new normalization is performed.

3. To see the correct angle tracking information, select **Automatic Measurements** from the **Operation** menu, or click the Automatic Measurements button from the Analysis toolbar.

Figure 4-16 Angle correction



4. To return to the stored nominal settings, click Return to Nominal icon.

Performing IRIS-specific Tasks

Once you have loaded or created a valid IRIS setup for the tube you need to inspect (see “Configuring Ectane IRIS Acquisition Setups” on page 100) and before performing a calibration, you need to validate your detection gate settings. The IRIS Application Wizard normally places gates very close to where they should be, but depending on the probe and turbine model you are using, you may need to perform some adjustments.

These adjustments are performed from an A-scan view:

- The target pin gate (gray) is used to detect the rotation synchronization signal generated by the target pin located on the turbine.
- The front wall gate (red) is used to detect the echo coming from the internal diameter of the tube, also called the front wall echo.
- The back wall gate (blue) is used to detect the echo coming from the external diameter of the tube, also called the back wall echo.

Preparing for Gate Adjustments

Before adjusting the gates, you need to perform a couple of operations:

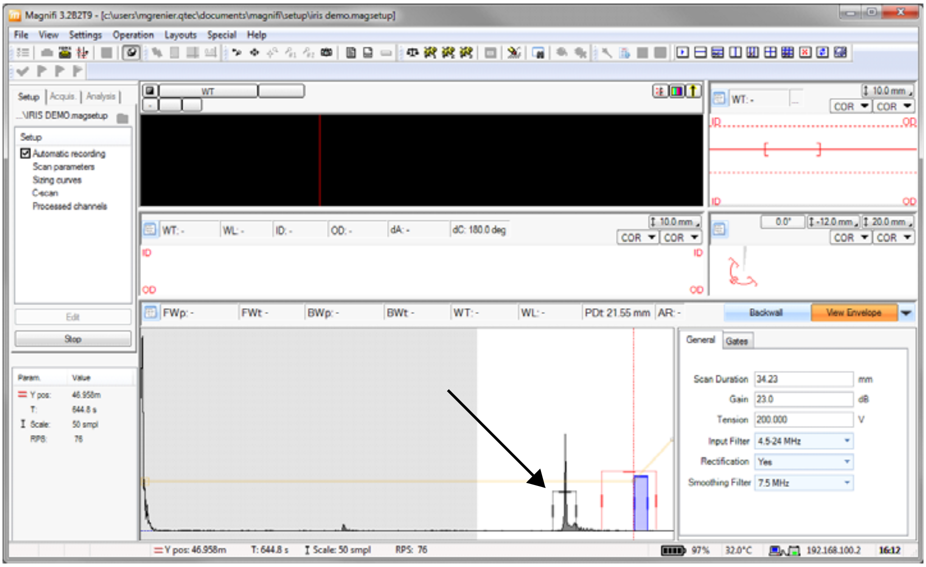
1. Display the A-scan view in your layout (see “Elements of an A-scan” on page 23).
2. Start the rotation of the turbine and submerge it in water.

Adjusting the Target Pin Gate

To adjust the target pin gate:

1. Start a Magnifi acquisition.
2. Click **View Envelope** (see “Elements of an A-scan” on page 23). Since the refresh rate of the A-scan is much lower than that of a C-scan, it may take a few seconds before the target pin signal starts to appear in the A-scan.
3. With the mouse, move the gray gate over the pin signal (see Figure 4-17).
4. Adjust the level of the gate to about 30% of the maximum signal amplitude, and the width slightly larger than the signal (see Figure 4-17).

Figure 4-17 Adjusting the target pin gate level



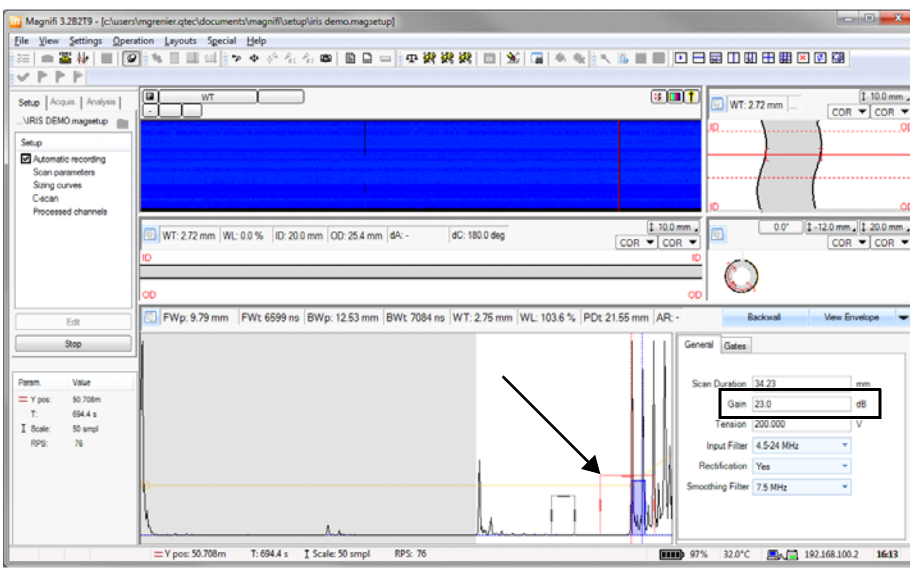
5. Once the target pin gate is set properly, the red cursor in the C-scan should start to move, and the rotation speed indicator should give a correct value.

Adjusting the Front Wall Gate

To adjust the front wall gate:

1. Click **View Envelope** to stop cumulating the A-scan.
2. Insert the probe in the tube. New echoes corresponding to the internal diameter of the tube will appear on the right-hand side of the A-scan.
3. With the mouse, move the red gate over the front wall signal (see Figure 4-18).
4. If needed, adjust the gain so that the front wall echo amplitude is about 100% of the screen height (slightly saturated) (see Figure 4-18).
5. Adjust the level of the gate to about 25% of the maximum signal amplitude (see Figure 4-18).
6. Probe centering is not always perfect. Because of that, echoes may move in position along the time axis of the A-scan. Adjust the width of the gate to make sure that the echo is properly detected, regardless of its position along the time axis.
7. Once the gate is set properly, the internal diameter should be well displayed in the projection view (see Figure 4-18).

Figure 4-18 Adjusting the front wall gate

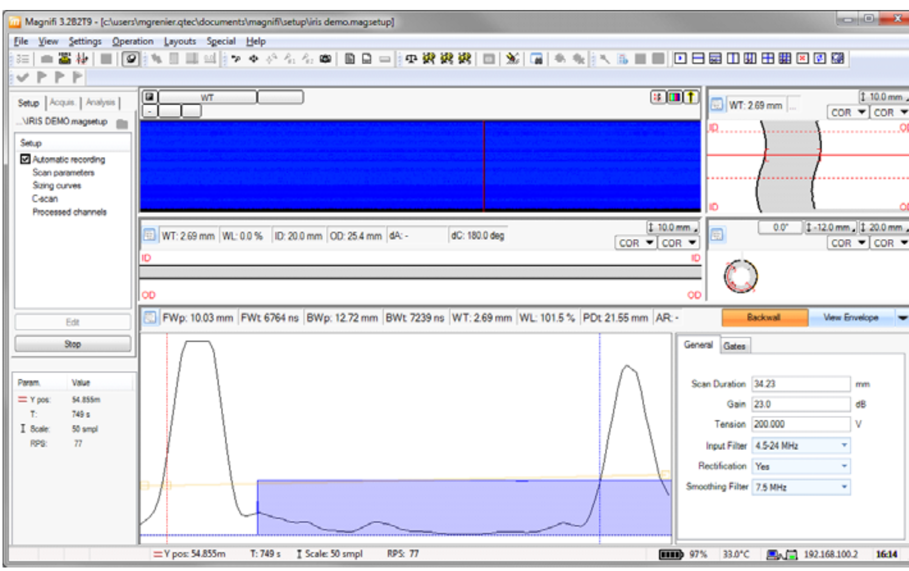


Adjusting the Back Wall Gate

To adjust the back wall gate:

1. Click **Backwall** in the header of the A-scan (see “Elements of an A-scan” on page 23). The first echo on the left side of the A-scan represents the front wall echo, and the second echo on the right side represents the back wall echo.
2. Change the TGC slope to increase or decrease the amplitude of the back wall echo. When possible, try to have an average amplitude on the back wall that is about 75% of the screen height. The TGC slope can be modified directly on the A-scan by moving the handles on the yellow line. You can also type a value in the **TGC Slope** field located in the **Gate** tab (see “Elements of an A-scan” on page 23).
3. With the mouse, adjust the levels of the blue gate. Try to set the level as low as possible (to detect small indication) without triggering on the noise.
4. Adjust the beginning of the gate as close as possible to the front wall echo without including it.
5. Once the gate is well set, the wall thickness of the tube should be well displayed in the projection view.

Figure 4-19 Adjusting the back wall gate

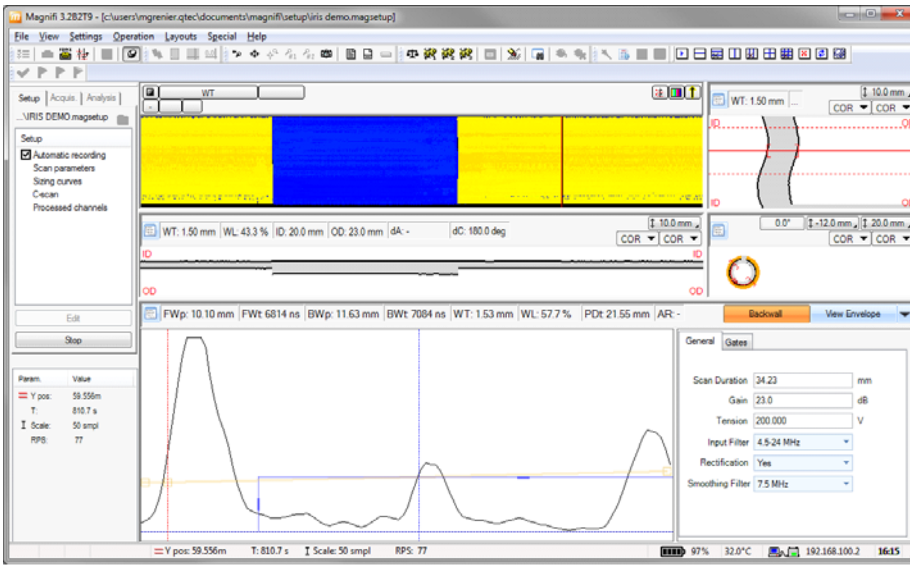


Validating Defect Detection

To do so:

1. Move the probe in the tube to validate that all defects are properly detected.
2. If needed, readjust the back wall gate to improve the detection of defects.

Figure 4-20 Validating defect detection



Calibrating Systems

Once your Ectane is properly connected and your setup is usable, you need to calibrate the system to ensure accurate results. You can calibrate from many data sources:

- Channels
- Channels and sizing curves
- Sizing curves
- C-scan
- C-scan and sizing curves
- IRIS
- Landmarks

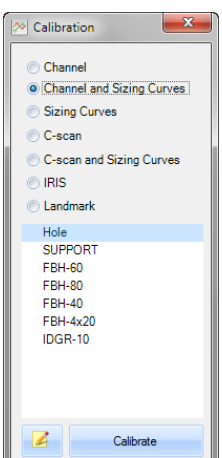
Calibrating Channels

To do so:

1. Load an existing setup (for more information, see “Opening Inspection Projects” on page 246).
2. Balance the probe on a sound area (see “Balancing” on page 225).
3. Acquire data from the calibration sample (block or tube).

4. Select **Operation>System Calibration** or click **System calibration** from the General toolbar. The **Calibration** window appears.

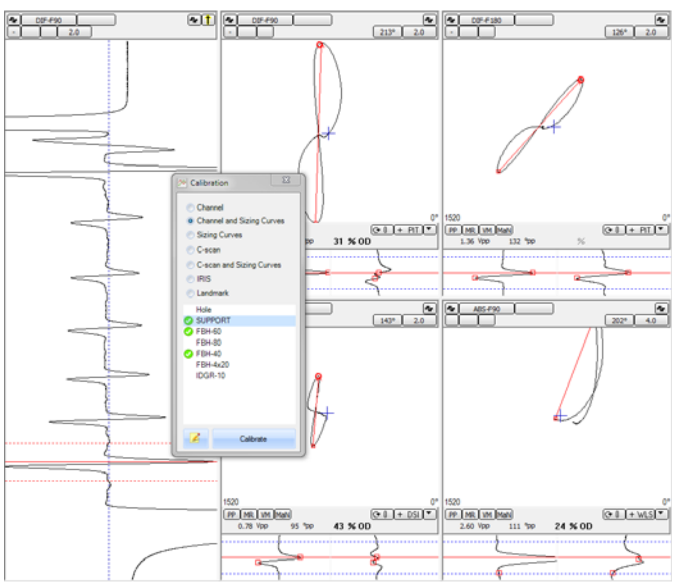
Figure 4-21 The **Calibration** window



5. Select **Channel** or **Channel and Sizing Curves:** this will adjust the rotation and gain of all channels that have been submitted to a Scale process (see “Setting Up Processed Channels” on page 142) and/or measure the references required to generate the sizing curve (see “Setting Up Sizing Curves” on page 200).
   If you select **Sizing Curves** only, it will perform measures similar to the other calibrations, but without changing the calibration itself (gain and rotation). It will only measure anew the sizing curves (see “Setting Up Sizing Curves” on page 200).
6. Select a reference in the list underneath the calibration radio buttons. The content of the list depends on the selection made in step 5.
7. Position the data selection cursor on the signal corresponding to the reference selected previously.

8. Click the measurement button. Once the reference measurement is taken, a green check mark appears next to the reference name in the list.
9. Repeat steps 6 to 8 for all references that you want to use for calibration purposes. You do not have to take measurements for all references.

Figure 4-22 Calibration—Channels and Sizing Curves



10. Click **Calibrate.** System calibration is performed based on the reference measurements taken.
11. Click **Close.**

Note *After calibration, it is always possible to rotate the signal further by click-dragging the rotation button in the Lissajous (see page 14.)*

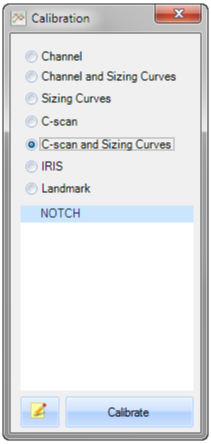
Calibrating C-scans

C-scan calibration is similar to channel calibration (previous section) except that the data is selected from the C-scans.

To do so:

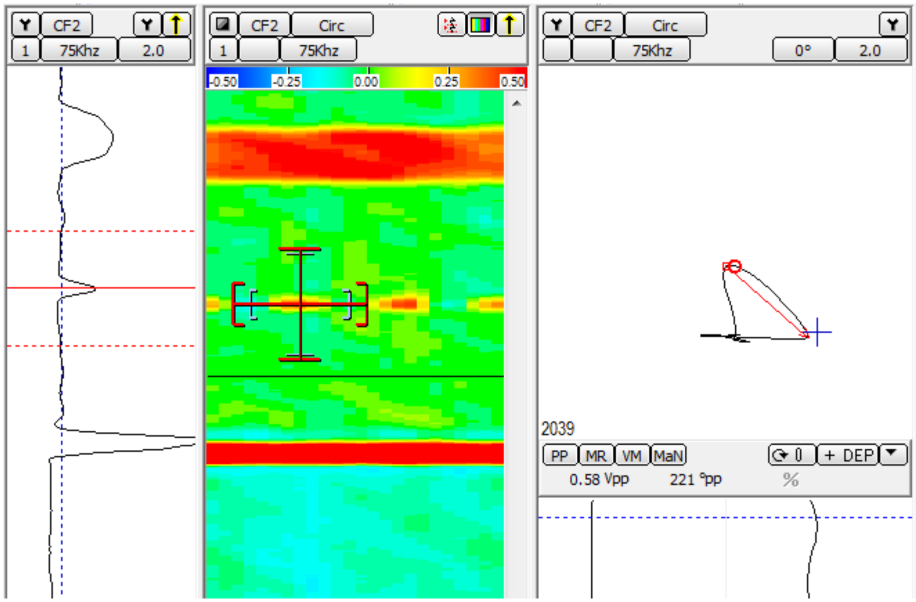
1. Select **Operation>System Calibration** or click **System calibration** from the General toolbar. The **Calibration** window appears.

Figure 4-23 The **Calibration** window



2. Select **C-Scan** or **C-Scan and Sizing Curves:** this will adjust the rotation and gain of the C-scans that have been submitted to a Scale process (see “Managing C-scans” on page 162) and/or measure the references required to generate the sizing curve related to a C-scan (see “Setting Up Sizing Curves” on page 200).
3. Select a reference in the list underneath the calibration radio buttons. The content of the list depends on the selection made in step 2.
4. Position the data selection cursor on the signal corresponding to the reference selected previously.

Figure 4-24 C-scan calibration



5. Click the measurement button. Once the reference measurement is taken, a green check mark appears next to the reference name in the list.

Note *The measurement is performed along the axis defined in the Scale process of the C-scan. If the measurement is along the Y axis, make sure that the vertical segment of the selection brackets is placed over the defect.*

6. Repeat steps 3 to 5 for all references that you want to use for calibration purposes. You do not have to take measurements for all references.
7. Click **Calibrate.** System calibration is performed based on the reference measurements taken.
8. Click **Close.**

Note *After calibration, it is always possible to rotate the signal further by click-dragging the rotation button in the Lissajous (see page 14.)*

Calibrating Landmarks

Landmark calibration is normally performed on representative data coming from heat exchangers. The default setup provided with Magnifi includes two landmarks named TS1 and TS2, which represent tubesheets at both ends of the heat exchanger. Different and/or additional landmarks could be present in your setup if you have changed the default Landmark table (see “Building Landmark Tables” on page 180).

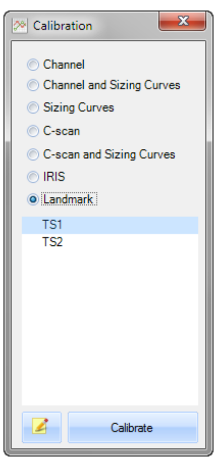
The purpose of this calibration is to associate a signal shape to each landmark that needs to be detected by Magnifi. During calibration, Magnifi evaluates the best way to automatically detect this signature so that every time a file is read, the landmark can be automatically placed in the code window.

To calibrate landmarks:

1. Acquire a tube from the exchanger or, if you are in analysis mode, read a file coming from the heat exchanger.
2. Temporarily change one strip chart and one Lissajous to show the channel named *Land*. This channel is automatically created by the Detect Landmark process (see “Editing the Detect Landmark Processing Unit” on page 157).

3. Select **Operation>System Calibration** or click **System calibration** from the General toolbar. The **Calibration** window appears.

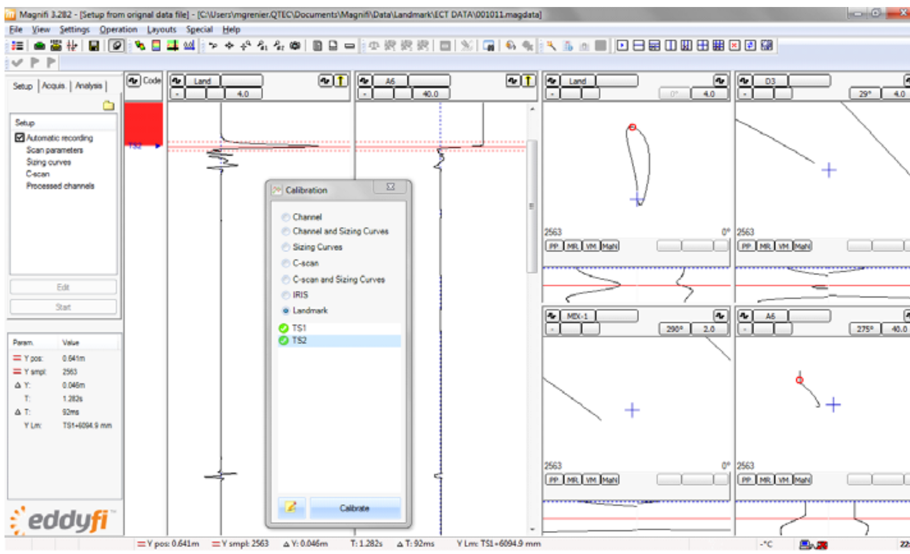
Figure 4-25 The **Calibration** window



4. Select **Landmark.** All landmark types present in the Landmark table should be listed.
5. Click on a landmark reference (in this example, TS1.)

6. Using the data selection cursor and the data selection brackets, select the signal shape in the *Land* channel strip chart that represents this landmark type.
7. Click the measurement button.
8. Repeat steps 5 to 7 for each landmark type that you have in the list.
9. Click **Calibrate**. System calibration is performed based on the reference measurements taken.
10. Click **Close.**

Figure 4-26 Landmark calibration



Calibrating IRIS

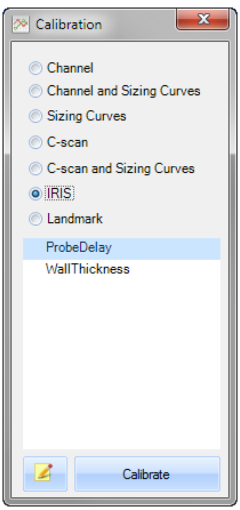
IRIS calibration is important to ensure that the calibration tube dimensions (internal and external diameters, wall thickness,...) are measured correctly.

To do so:

1. Load an existing setup (for more information, see “Opening Inspection Projects” on page 246).
2. Acquire data from the calibration tube.
3. Move the C-scan data selection cursor on a sound region of the tube, which represents the nominal condition.

4. Select **Operation>System Calibration** or click **System calibration** from the General toolbar. The **Calibration** window appears.

Figure 4-27 The **Calibration** window



5. Select **IRIS**.

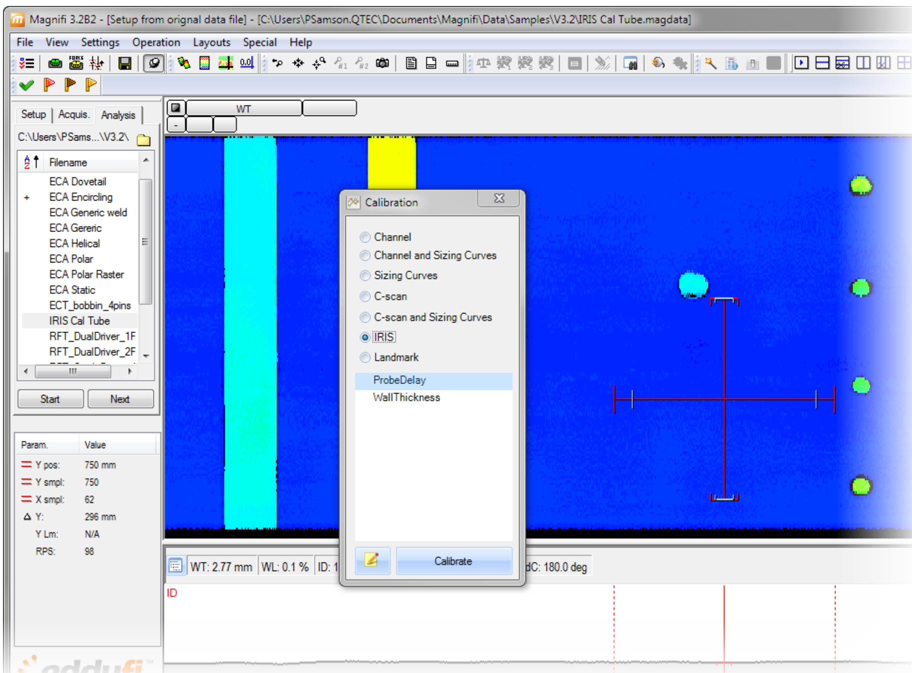
6. Select **ProbeDelay** and click the measurement button. Once the reference measurement is taken, a green check mark appears next to the reference name in the list.
   This measurement will evaluate the internal diameter around the circumference at the cursor location and recalculate the probe delay so that the average internal diameter matches the nominal internal diameter defined in the Application Wizard.

7. Select **WallThickness** and click the measurement button. Once the reference measurement is taken, a green check mark appears next to the reference name in the list.

This measurement will evaluate the wall thickness around the circumference at the cursor location and recalculate the material ultrasound velocity so that the average wall thickness matches the nominal wall thickness defined in the Application Wizard.

8. Click **Calibrate.** System calibration is performed based on the reference measurements taken.
9. Click **Close.**

Figure 4-28 IRIS calibration



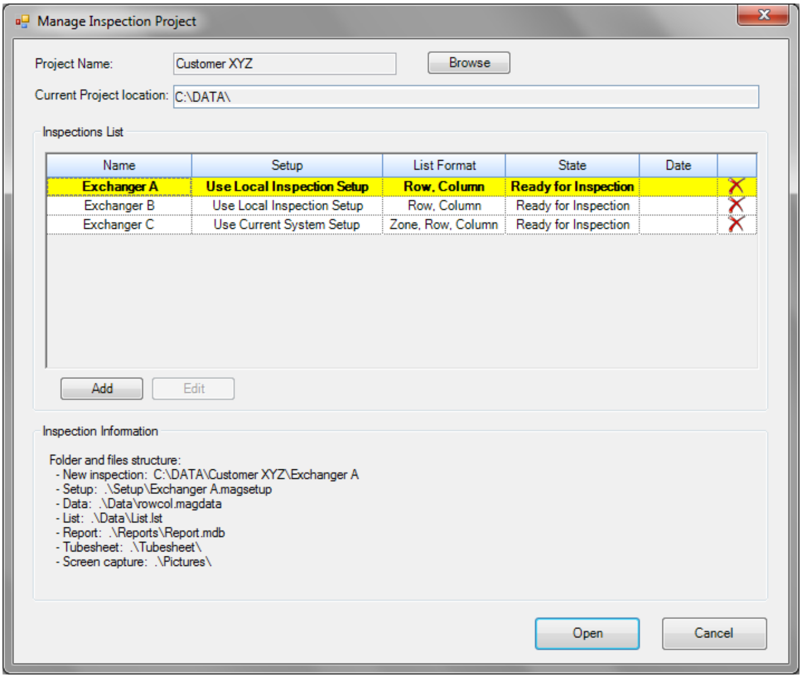
Opening Inspection Projects

Usually, after installing your inspection equipment and establishing communication with it, the first step to perform is to open an inspection project in Magnifi.

To do so:

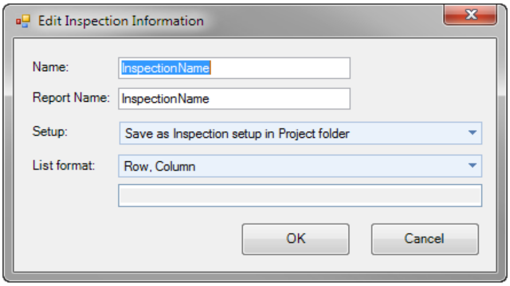
1. From inside Magnifi, select **File>Manage Inspection Project.** The **Manage Inspection Project** window appears.

Figure 4-29 The **Manage Inspection Project** window



2. If the project opened by default is not the one you are looking for, click **Browse** next to the **Project Name** text box. From the **Open** window that appears, select the inspection project that you want to open and click **OK.** If you do not have an inspection project defined for your inspection, see "Creating Inspection Projects" on page 214.
3. In the list of inspections displayed, select the appropriate inspection.
4. If you need to perform a new inspection for the same project, click **Add** to define a new inspection in the **Inspections List.** The **Edit Inspection Information** window appears.

Figure 4-30 The **Edit Inspection Information** window



5. Enter the inspection name in the **Name** text box.
6. Enter the report name in the **Report Name** text box.
7. In the **Setup** drop-down list, select how you want to manage the inspection setup file:
   - **Save as Inspection Setup in Project Folder:** (default value) copy and rename the current setup and saves it under the Setup directory.

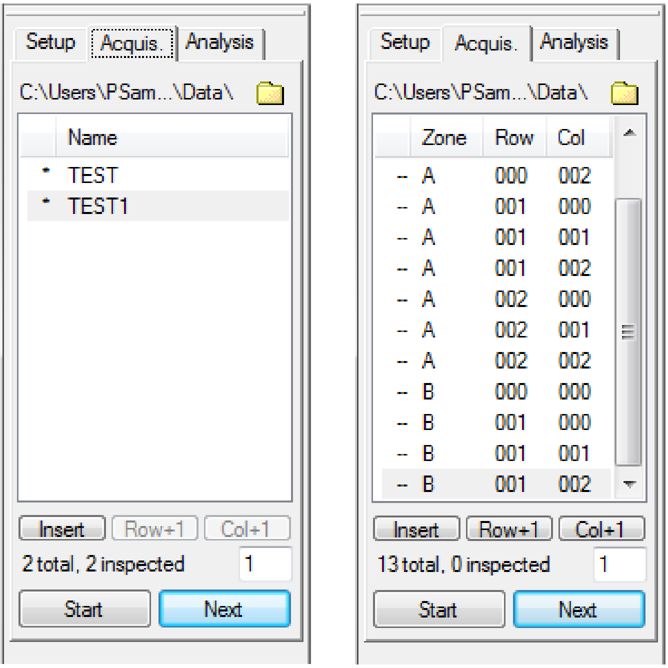
Important When you use this option, you must already have the appropriate setup loaded in Magnifi.

   - **Keep current name and location:** performs no action on the setup file. The current name is kept and the file is not copied in the Setup directory. This is the option to use if you prepare in advance several inspections and the current Magnifi setup is not necessarily the one to be used for this inspection.
8. In the **List format** drop-down list, select the list format that you want to use:
   - **Row, Column**
   - **Zone, Row, Column**
   - **Free format**
   - **Import**
     This last option allows you to select a *.*lst* file, using the **Browse** button. This file will be copied in the Data folder of the inspection project directory, and renamed *List.lst*. Its name and location will be displayed in the text box underneath the **List format** drop-down list.
9. Click **OK.** The inspection is loaded with the proper setup, tube list format and report. The acquisition and analysis folder are selected at the same time.

Creating Inspection Lists

Before performing acquisitions, it is necessary to create an inspection list. The format of the inspection list is defined when you create or open an inspection project (see “Opening Inspection Projects” on page 246).

Figure 4-31 The different looks of an inspection list



In Freeform Format

To create an inspection list:

1. From the **Acquis.** tab, click **Insert.** A first row is created.
2. Name the first row. You are now ready to start inspection. After starting and stopping the first acquisition, a new row is automatically created when you click **Next.**

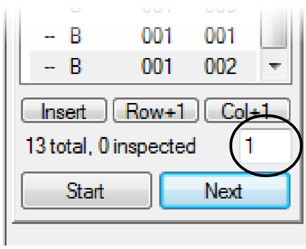
In (Zone), Row, Column Format

To create an inspection list:

1. From the **Acquis.** tab, click **Insert.** A first row is created. The first row always starts at 0. Usually, users that work with rows and columns tend to create all their rows first.
2. Add rows by clicking **Row+1**.
3. Go back to each row and add columns by clicking **Col+1.**
4. If you are working in **Zone, Row, Column** format, you can add a new zone by clicking **Insert** again.

Note *To increase the row and column numbers by more than one, change the number under the* ***Col+*** *button to any interval that you want.*

Figure 4-32 Interval text box



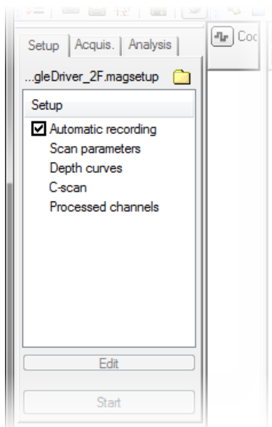
Once you have created your inspection list, you are ready to start performing acquisitions.

Performing Acquisitions

Once your system is balanced and calibrated, you are ready to start performing acquisitions on a larger scale.

Data acquired can be recorded in automatic mode, or in user-activated mode. By default, **Automatic recording** is selected in the **Setup** tab.

Figure 4-33 Automatic recording activated



To perform acquisitions in automatic recording mode:

1. Switch to acquisition mode by clicking the **Acquis.** tab in the Control bar, or by selecting **Acquisition** from the **Operation** menu.
2. Select a file from the file list in the **Acquis.** tab.
3. Once you have selected a file, you can start acquisitions in one of three ways:
   - By clicking **Start** on the **Acquis.** tab of the Control bar,
   - By selecting **Start** from the **Operation** menu, or
   - By pressing F2.

   Data starts appearing in the different views as you scan. The background appears in green, meaning that data is *saved automatically* to the hard drive.

Note *During an acquisition, you can clear the Lissajous by selecting **Clear Lissajous** from the **Operation** menu, or by pressing F5.*

4. Once you are done scanning, click **Stop** from the Control bar, select **Stop** from the **Operation** menu, or press F2 again. Data is processed as per your setup, and then displayed.
5. To start a new acquisition, click **Next** from the **Acquis.** tab of the Control bar, or press F4, and start recording the next file in the list.

6. Repeat steps 3 to 5 until all your acquisitions are performed. Data are saved automatically in the proper inspection project directories, as explained on page 246.

To perform acquisitions in user-activated recording mode (**Automatic recording** *not* selected):

1. Switch to acquisition mode by clicking the **Acquis.** tab in the Control bar, or by selecting **Acquisition** from the **Operation** menu.
2. Select a file from the file list in the **Acquis.** tab.
3. Once you have selected a file, you can start acquisitions in one of three ways:
   - By clicking **Start** on the **Acquis.** tab of the Control bar,
   - By selecting **Start** from the **Operation** menu, or
   - By pressing F2.

   Data starts appearing in the different views as you scan. The background appears in white, meaning that data *IS NOT* saved to the hard drive.
4. Click **Start/Save** from the Control bar or press F10. The background now appears green, meaning that data is now saved to the hard drive.
5. Click **Stop/Save,** or press F10 again to stop saving data. The background switches to white, meaning that data is not saved anymore (but still displayed on-screen.
6. Click **Stop,** or press F2 to stop the acquisition.
7. To start a new acquisition, click **Next** from the **Acquis.** tab of the Control bar, or press F4, and start recording the next file in the list.
8. Repeat steps 3 to 5 until all your acquisitions are performed. Data are saved automatically in the proper inspection project directories, as explained on page 246.

Displaying the Large Acquisition Window

The oversized Start Acquisition window is very useful when acquisitions are performed by a single operator. It allows you to see in a bigger font the name of the file that needs to be recorded. It also gives you the option of moving automatically to the next file when the acquisition is stopped.

To work with this window:

1. From the **Operation** menu, select **Large Acquisition Window.**
2. Resize the window to your liking and position it wherever you want on the screen. Everything in the window scales as you make the window larger or smaller.
3. Check the **Move to next file after acquisition stop** box if necessary.
4. Press F2 to start acquiring data.

Adding Notes

Magnifi allows you to add various notes to your inspections. You have the:

- **Acquisition Notepad,** where you can add notes to whole inspection project data files,
- Comments, where you can add a *.txt* file to a specific acquisition file.
- Bubble notes, that can display comments visually on screen, for reporting purposes. Bubble notes are covered in the next chapter (see "Analyzing & Reporting Data" on page 255).

On the Acquisition Notepad

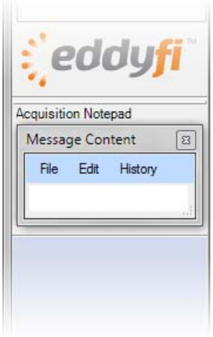
Notes on the **Acquisition Notepad** apply to all files in the acquisition list (**Acquis.** tab). These notes can be useful when something special happens at one point during a series of acquisitions (i.e., when a probe, equipment, or operator is changed, when there is noise coming from the environment, etc.) and you want to make a note of it for the analyst who will work on the files afterwards.

The acquisition notepad is displayed as a toolbar under the Eddyfi logo. To toggle the display of the **Acquisition Notepad,** select **Acquisition Notepad** from the **View** menu.

To enter notes on the **Acquisition Notepad:**

1. Click in the **Acquisition Notepad.** A small **Message Content** window opens.

Figure 4-34 The **Message Content** window in the **Acquisition Notepad**



2. Enter your note in the **Message Content** window.
3. Click the Close button. The **Message Content** window closes, and the message is now displayed in the **Acquisition Notepad.** This note will be attached to all subsequently recorded files.

Commenting on Specific Files

Comments are *.txt* files attached to a specific acquisition file in the acquisition list (**Acquis.** tab). That text file is automatically given the same name as the acquisition file to which it is attached. These comments are not associated with other acquisition files. The comments are visible in Analysis mode.

To comment on a specific file:

1. Select a file in the acquisition list (**Acquis.** tab).

2. Select **Add File Comments** from the **Operation** menu or click the Add File Comments button. A **Message Content** window opens in the centre of your screen.

Figure 4-35 The **Message Content** window.

3. Enter the comments about this specific file.
4. Click the Close button. The **Message Content** window closes.

Analyzing & Reporting Data

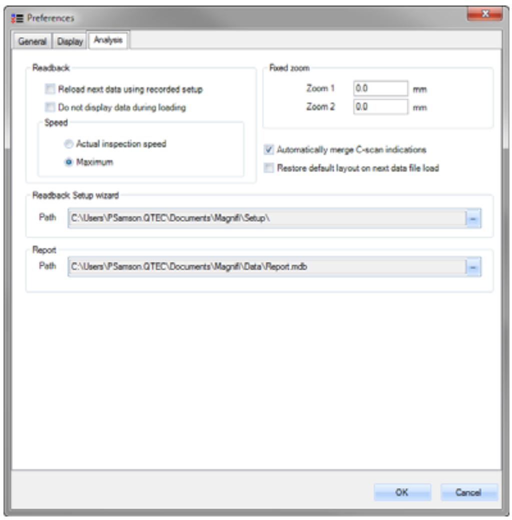
5

Setting Analysis Preferences

To do so:

1. From the **Settings** menu, select **Preferences.** You can also click **Preferences** from the General toolbar. The **Preferences** window appears.
2. Click the **Analysis** tab.

Figure 5-1 The **Analysis** tab



3. In the **Readback** section:
   - **a** Check the **Reload next data using recorded setup** box if you want to use the same setup file for all read back data.
   - **b** Check the **Do not display data during loading** box if you do not want to display data as it is loaded from memory.
   - **c** In the **Speed** subsection, select whether you want to display data at the speed it was acquired, or as fast as possible.

4. Click the browse button in the **Readback Setup wizard** section to browse to the folder where the setup files are stored.

5. Click the browse button in the **Report** section to browse to the folder where the reports are stored.
6. To automatically merge C-scan indications, check the **Automatically merge C-scan indications** box.
7. To bring back the default layout every time you read the next file in a series, check the **Restore default layout on next data file load** box.

For information on zooms, see “Working with Zooms” on page 261.

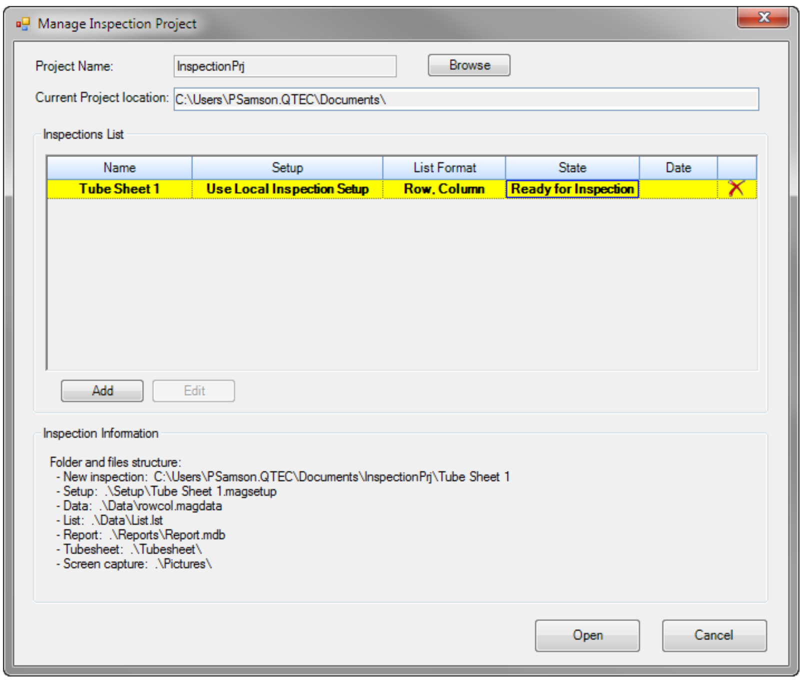
Opening Inspection Projects

Usually, the first step to perform, after switching to Analysis mode (see “Switching between Operation Modes” on page 8), is opening an inspection project in Magnifi.

To do so:

1. From inside Magnifi, select **File>Manage Inspection Project.** The **Manage Inspection Project** window appears.

Figure 5-2 The **Manage Inspection Project** window



2. If the project opened by default is not the one you are looking for, click **Browse** next to the **Project Name** text box. From the **Open** window that appears, select the inspection project that you want to open.
3. In the **Inspections List,** select the inspection that you want to analyze and click **Open.** The inspection opens with the appropriate analysis folder selected.

Loading Setup Files

When ready to analyze data, you need to load a setup file. There are three different ways to load a setup file:

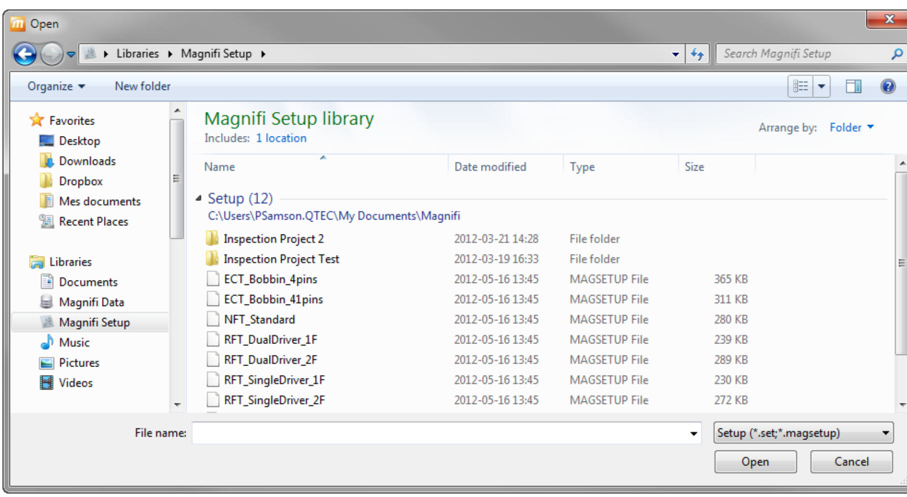
- By loading an existing setup file;
- By loading a recently used setup file;
- By loading the *original* setup file.

However, before loading a setup file, you should make sure that your general analysis options are to your liking (see “Setting Analysis Preferences” on page 256).

Loading Existing Setup Files

To load an existing setup, select **Load Setup** from the **File** menu. A window opens, where you can select from many existing setups. You can also browse through directories to find the setup file that you want.

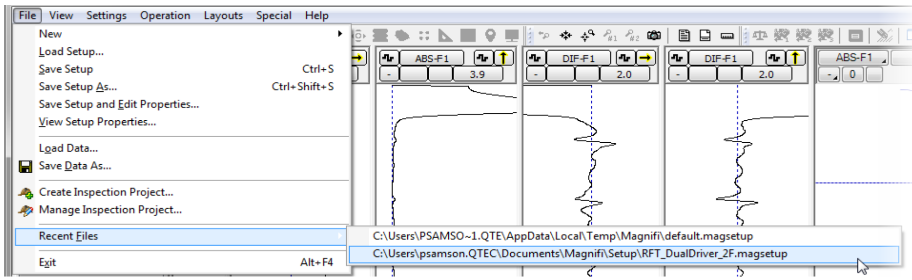
Figure 5-3 Loading an existing setup file



Loading Recently Used Setup Files

If you recently used a setup file, you can quickly access it by selecting it from the **Recent Files** item in the **File** menu.

Figure 5-4 Selecting a recent file



Loading *Original* Setup Files

When loading a data file from the **Analysis** tab, you can load the original setup recorded with this data by making sure that the Readback with current setup button is *not* selected.

Note *This button is selected by default.*

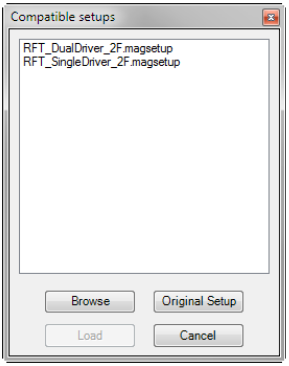
When loading the file while this button is *not* selected, Magnifi will use the setup that you had when the data was recorded. Once the file is loaded, the Readback with current setup button is automatically selected so that you keep this setup the next time that you read the file

Reading Data Files

To read data files:

1. Select a file from the **Analysis** tab.
2. Click **Start,** or press F2 to read the file. If the data is not compatible with the currently loaded setup, the **Compatible setups** window appears.

Figure 5-5 The **Compatible setups** window



From this window, you can:

- Apply the original setup by clicking **Original Setup.**
- Select one of the displayed setup and click **Load.**
- Browse through your directories to find a compatible setup by clicking **Browse.**

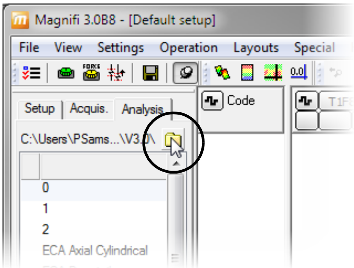
Changing Saved Data Directories

When opening an inspection project, a data directory is already defined for data gathered during the acquisition.

If you want to analyze data found in another directory:

1. Click the folder icon in the top right corner of the **Analysis** tab. The **Select Directory** window appears.

Figure 5-6 Clicking on the folder icon



2. Browse to the directory where you will find data to analyze.
3. Click **OK.** The **Select Directory** window closes and all files available in this directory for analysis appear in the **Analysis** tab.

Measuring Indications

When performing data analysis, one of the most important operations consists in measuring signals that could be a defect indication.

We can separate the measurement process in two steps:

- Data selection
- Measurement

Positioning Data for Selection

When selecting data, Magnifi offers two additional ways of accessing the displayed data: centering and *best-fitting*.

When centering, Magnifi centers the displayed data based on the position of the data selection cursor.

When *best-fitting,* Magnifi centers the data AND adjusts the scale so that the maximum amplitude signal found within the data selection brackets reaches 75% in the Lissajous.

Working with Zooms

Magnifi allows you to set predefined zoom levels that can later be accessed from the Analysis toolbar.

To do so:

1. From the **Settings** menu, select **Preferences.** The **Preferences** window appears.
2. Click the **Analysis** tab (see Figure 5-1).
3. In the **Fixed zoom** section, enter different scales, (in **mm** or **in.**) in the **Zoom 1** and **Zoom 2** boxes.
4. Click **OK.** The fixed zooms are set.

To quickly access the set zoom levels, click the Fixed Zoom 1 or Fixed Zoom 2 buttons in the Analysis toolbar.

You can also go back to the previous zoom level by clicking the Previous zoom button.

Selecting Data in Strip Charts

Note *For more information on view interface elements, see “Elements of a Lissajous” on page 15, or see “Elements of a Voltage Plane View” on page 18.*

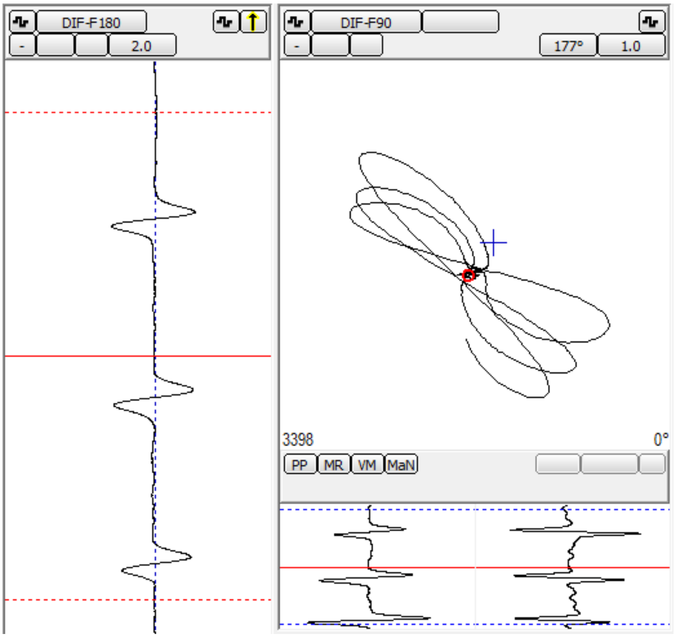
The data selection is done with the data selection cursor located in the strip chart view (red cursors).

To select data:

1. Click in the strip chart view to automatically bring the data selection cursor to the location of the click. All the data contained within the data selection brackets (red dotted lines) are represented in the Lissajous or voltage plane view.
2. Click and drag the selection brackets to reduce or increase the size of the selection.

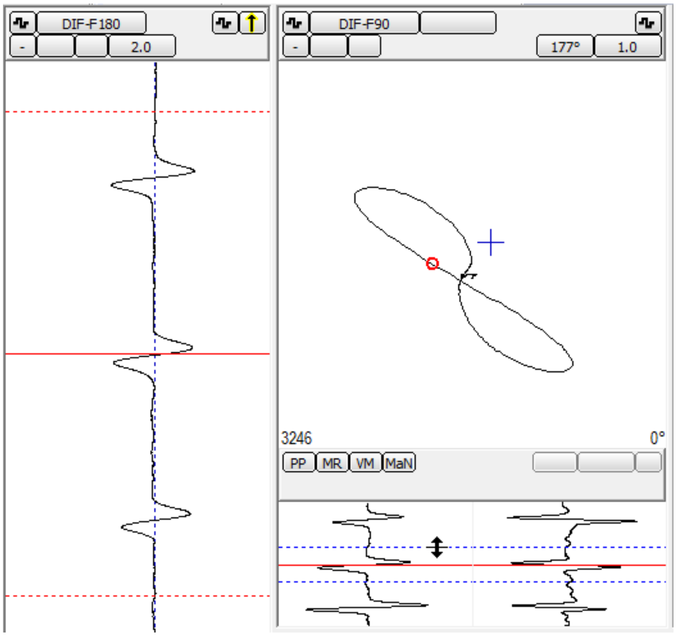
Note *For a more accurate selection, Magnifi offers various ways to zoom in and out. For more information, see “Working with Zooms” on page 261.*

Figure 5-7 Selecting data in the strip chart



In the Lissajous view, it is also possible to adjust the selection size by moving the measurement brackets (blue) located in the mini-strip without changing the size of the data selection brackets.

Figure 5-8 Selecting data in the mini strip



If the data selected appears too small or too big in the Lissajous or voltage plane view, you can change the scale value by left- or right-clicking the Scale button. A manual scale can also be made by right-clicking and dragging up or down in the data area of the view.

Measuring Data in Lissajous and Voltage Plane

Note *For more information on view interface elements, see "Elements of a Lissajous" on page 15, or see "Elements of a Voltage Plane View" on page 18.*

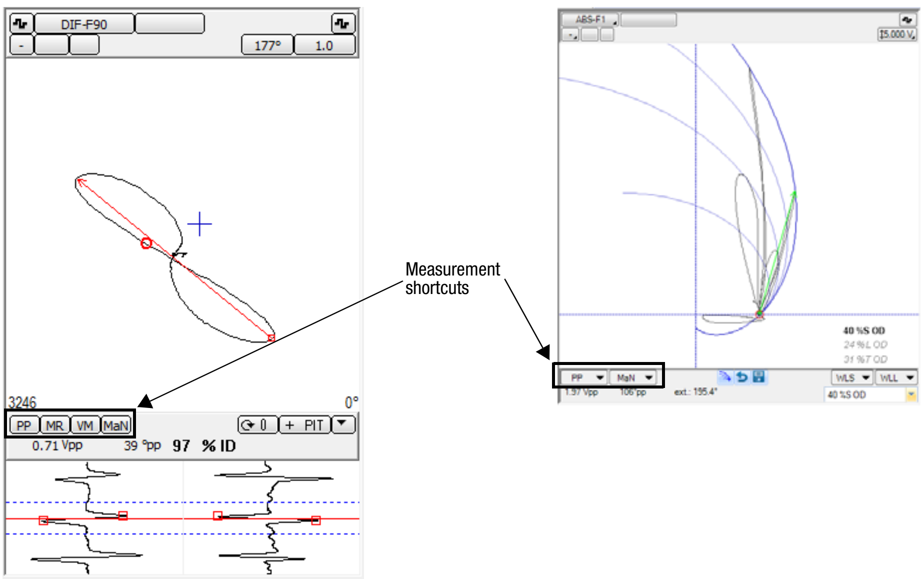
There are three ways of enabling the measurement tools:

- By using the Automatic measurement function
- By using measurement shortcuts
- By using contextual measurements

To measure data:

1. Click Automatic Measurement or select **Automatic Measurements** from the **Operation** menu.
   - **a** Automatic measurement is performed on the selected data. The voltage and angle value are based on the **Default measurement method** defined in the **General** tab of the **Preferences** (see “Setting Amplitude/Phase Measurement Methods” on page 74).
   - **b** A vector representing the measurement is then displayed in the Lissajous or voltage plane view.
   - **c** The calculated signal amplitude and angle are displayed in the measurement bar of the Lissajous or voltage plane display.

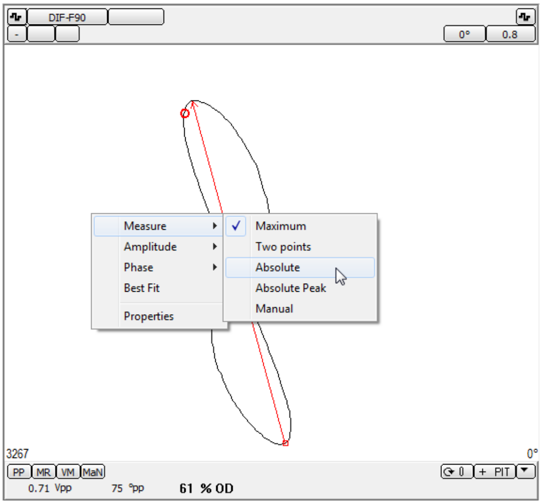
Figure 5-9 Calculated signal and angle



2. Click the Measurement shortcut at the bottom of the Lissajous or voltage plane. Measurement is performed on the *selected* data. The shortcuts give you additional measurement methods.

   In voltage plane view, it is even possible to define the measurement mode that you want to have on the shortcut button by clicking the arrow located on the bottom right side of the button.
3. Right-click in the Lissajous or voltage plane view. A contextual menu appears and allows you to select the measurement mode of your choice.

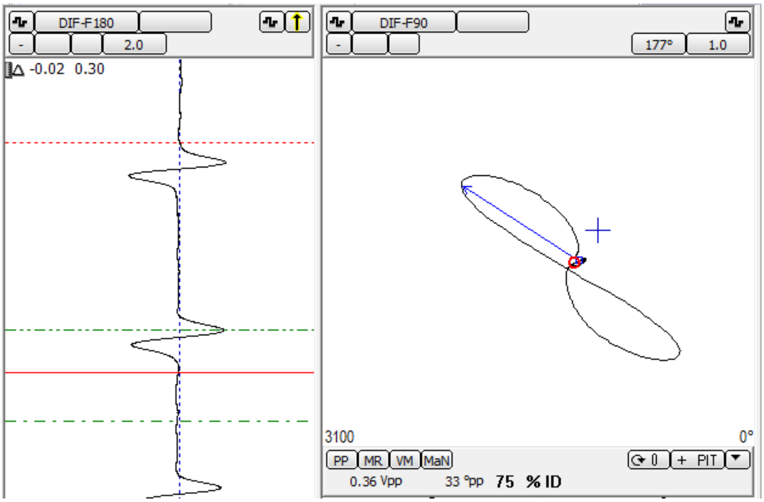
Figure 5-10 Selecting a measurement mode from the contextual menu



You can select the **Measure** mode:

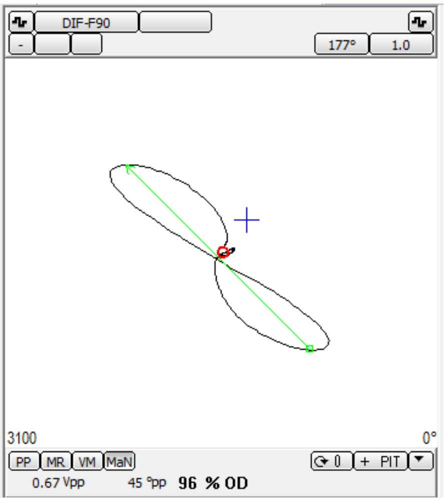
- **Maximum:** Magnifi automatically displays the measurement vector (red) and calculates the corresponding amplitude and angle.
- **Two-points:** activates two additional cursors in the strip chart. The measurement vector (blue) is based on the data located at the position of the two cursors, without considering other points between them.

Figure 5-11 Measuring with two points



- **Absolute:** uses a straight line from the zero position to the main cursor position.
- **Absolute Peak:** uses a straight line from the zero position to the peak value of the data selection.
- **Manual:** allows you to use the mouse to manually trace a vector in the Lissajous. The amplitude and angle values are calculated based on the vector (green) drawn on screen.

Figure 5-12 Measuring manually



You can also select amplitude and phase measurements mode from the contextual menu (see Figure 5-10). For more information on the various measurements available for these measurements, see “Setting Amplitude/ Phase Measurement Methods” on page 74.

Selecting and Measuring Data in IRIS Projections

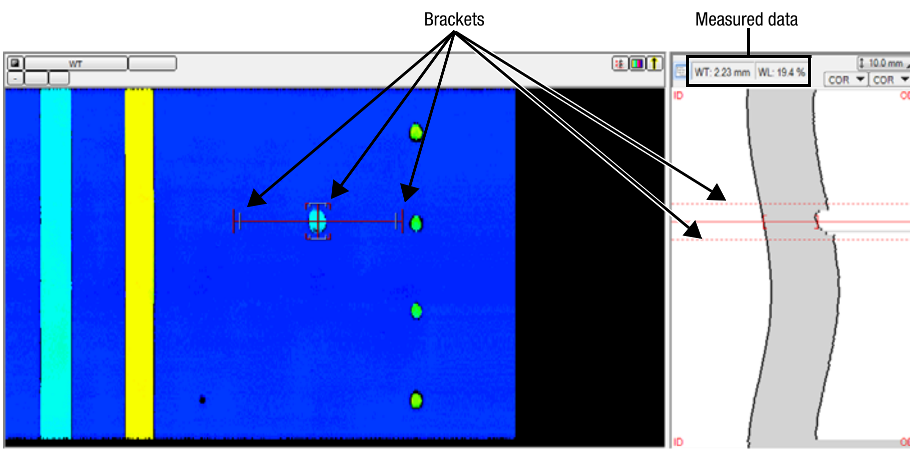
Note *For more information on view interface elements, see “Elements of a Projection View” on page 29.*

The data selection is done in either the C-scan or a projection view (except cylindrical projection view).

To select data, simply click on an indication in the C-scan or projection view.

The data cursor automatically measures all data selected to appear in the information fields. You can modify the measured data by manually moving the various brackets, if necessary.

Figure 5-13 Selecting data in IRIS inspections



Using Screen Captures

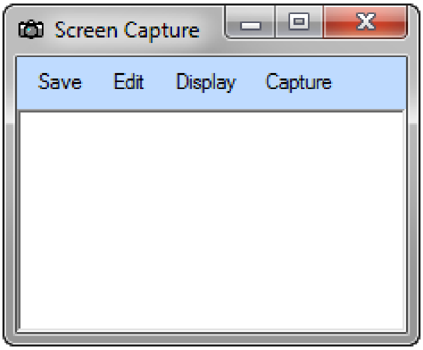
Magnifi comes with a tool that allows you to take screen captures of your application directly from within the application itself. You can save those screen captures to add in reports or print them directly from Magnifi.

Making Screen Captures

To make a screen capture:

1. From the **Operation** menu, select **Screen Capture.** The **Screen Capture** window appears.

Figure 5-14 The **Screen Capture** window



Note *The screen capture tool is only compatible with Windows 7 Basic color schemes. The workstation is automatically placed in a compatible color scheme whenever the screen capture tool is selected. It reverts to the original color scheme once you close the screen capture tool.*

2. From the **Capture** menu, select the screen element that you want to capture:
   - **Screen:** captures the entire screen without further operations.
   - **Window:** captures the window or window element on which you click.
   - **Rectangle:** captures a rectangular area that you select.
3. If you selected **Window** or **Rectangle,** make your selection on the screen. The selected screen capture appears as a preview in the **Screen Capture** window.

Saving Screen Captures

Once you have made a screen capture, you can save it for use in another application or document.

To do so:

1. With the screen capture previewed in the **Screen Capture** window, select **Save** from the **File** menu. A standard **Save image** window appears.
2. Browse to the directory where you want to save the screen capture.
3. In the **File name** text box, enter the name under which you want to save your screen capture.

Note *Image files created with the **Screen Capture** tool can only be saved in .bmp format.*

4. Click **Save.** The screen capture is saved in the directory you chose, under the name you gave it.

Printing Screen Captures

Once you have made a screen capture, you can print it directly from Magnifi.

To do so:

1. With the screen capture previewed in the **Screen Capture** window, select **Print** from the **File** menu. The **Print** window for your default printer appears.
2. Configure your print job as necessary and click **OK.** The screen capture is printed to your default printer.

Closing the Screen Capture Tool

Once you have performed your screen capture tasks, you can close the Screen Capture tool by selecting **Close** from the **File** menu, or by clicking the Close box in the top right corner of the window.

Note *If you normally use a color scheme that is not compatible with Windows 7 Basic, you revert to that color scheme on closing the Screen Capture tool.*

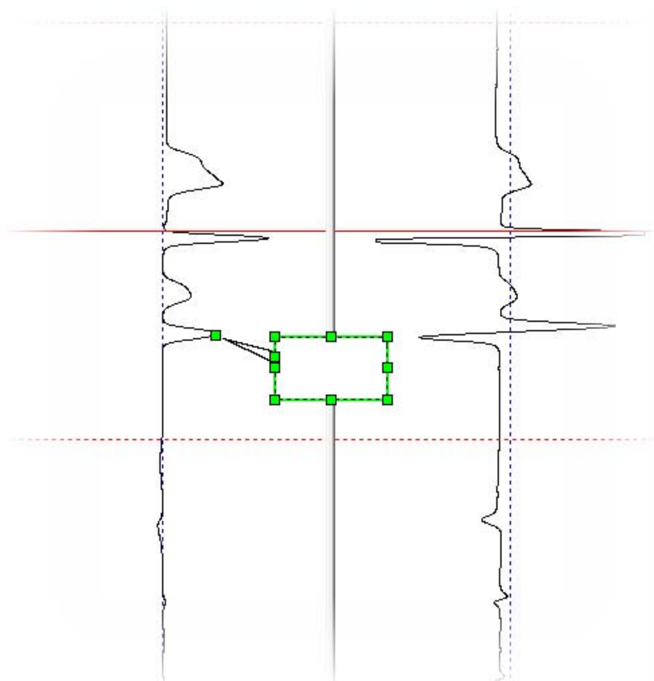
Creating Bubble Notes

When using screen captures, you can add visual notes to the document.

To do so:

1. From the **Operation** menu, select **Create Note.** A bubble note appears in the data display area.

Figure 5-15 A bubble note on screen



2. Use the green positioning handles to make the bubble note point to the screen element of your choice.
3. Enter text and click outside the bubble note.

Important These notes cannot be saved and are only usable on screen captures. Once you close the document, the notes disappear and cannot be retrieved.

Managing Reports

Once you have performed your scans and acquired data from your inspections, you can generate reports to keep track of your inspections.

Configuring Reports

The first thing that you need to do when producing reports is to decide which data appears in the report. You can do so in one of two ways:

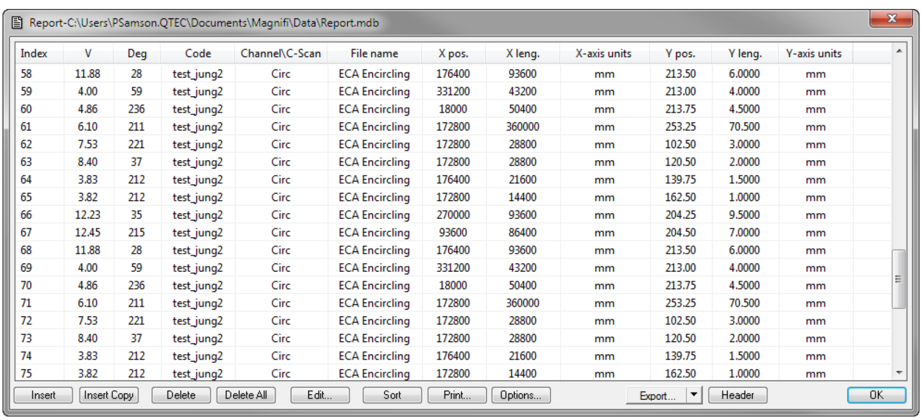
- From predefined profiles
- From scratch

From Predefined Profiles

To do so:

1. Select **Edit Report** from the **Operation** menu, or click **Edit report** from the **Analysis** toolbar. The **Report** window opens, and the path to the file is indicated in the title bar.

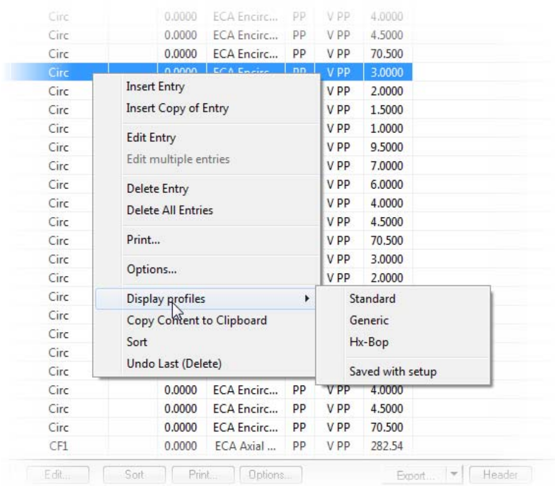
Figure 5-16 The **Report** window



Index	V	Deg	Code	Channel\C-Scan	File name	X pos.	X leng.	X-axis units	Y pos.	Y leng.	Y-axis units
58	11.88	28	test_jung2	Circ	ECA Encircling	176400	93600	mm	213.50	6.0000	mm
59	4.00	59	test_jung2	Circ	ECA Encircling	331200	43200	mm	213.00	4.0000	mm
60	4.86	236	test_jung2	Circ	ECA Encircling	18000	50400	mm	213.75	4.5000	mm
61	6.10	211	test_jung2	Circ	ECA Encircling	172800	360000	mm	253.25	70.500	mm
62	7.53	221	test_jung2	Circ	ECA Encircling	172800	28800	mm	102.50	3.0000	mm
63	8.40	37	test_jung2	Circ	ECA Encircling	172800	28800	mm	120.50	2.0000	mm
64	3.83	212	test_jung2	Circ	ECA Encircling	176400	21600	mm	139.75	1.5000	mm
65	3.82	212	test_jung2	Circ	ECA Encircling	172800	14400	mm	162.50	1.0000	mm
66	12.23	35	test_jung2	Circ	ECA Encircling	270000	93600	mm	204.25	9.5000	mm
67	12.45	215	test_jung2	Circ	ECA Encircling	93600	86400	mm	204.50	7.0000	mm
68	11.88	28	test_jung2	Circ	ECA Encircling	176400	93600	mm	213.50	6.0000	mm
69	4.00	59	test_jung2	Circ	ECA Encircling	331200	43200	mm	213.00	4.0000	mm
70	4.86	236	test_jung2	Circ	ECA Encircling	18000	50400	mm	213.75	4.5000	mm
71	6.10	211	test_jung2	Circ	ECA Encircling	172800	360000	mm	253.25	70.500	mm
72	7.53	221	test_jung2	Circ	ECA Encircling	172800	28800	mm	102.50	3.0000	mm
73	8.40	37	test_jung2	Circ	ECA Encircling	172800	28800	mm	120.50	2.0000	mm
74	3.83	212	test_jung2	Circ	ECA Encircling	176400	21600	mm	139.75	1.5000	mm
75	3.82	212	test_jung2	Circ	ECA Encircling	172800	14400	mm	162.50	1.0000	mm

2. Right-click anywhere in the report. A contextual menu appears.
3. From the contextual menu, select **Display profiles.**

Figure 5-17 Selecting **Display Profiles** from the contextual menu



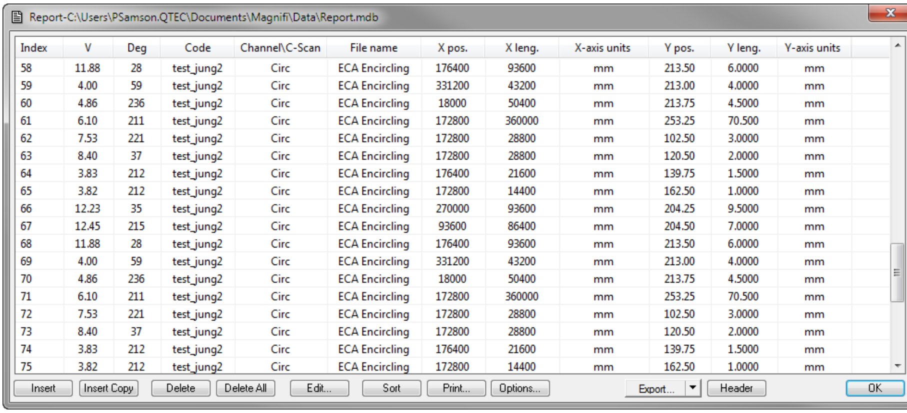
4. In the **Display profiles** sub-menu, select one of the following options:
   - **Standard:** columns displayed are relevant to general tube inspections
   - **Generic:** columns displayed are more relevant to array inspections
   - **Hx-Bop:** columns displayed are similar to the Standard profile, but have no information on analysts
5. Once the report profile is displayed, you can customize it as explained in the next section.

From Scratch

To do so:

1. Select **Edit Report** from the **Operation** menu, or click **Edit report** from the **Analysis** toolbar. The **Report** window opens, and the path to the file is indicated in the title bar.

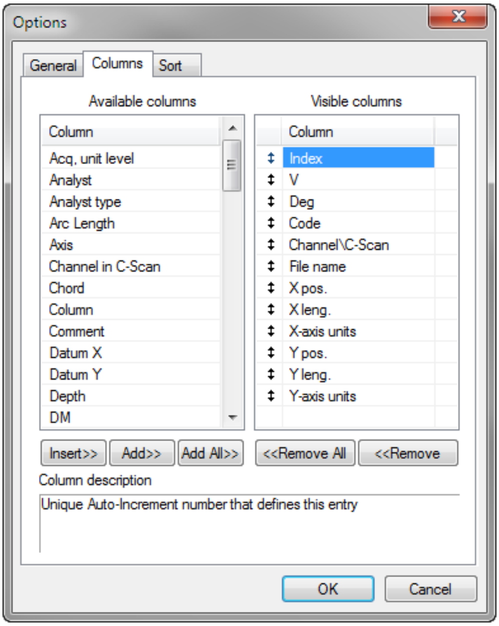
Figure 5-18 The **Report** window



Index	V	Deg	Code	Channel\C-Scan	File name	X pos.	X leng.	X-axis units	Y pos.	Y leng.	Y-axis units
58	11.88	28	test_jung2	Circ	ECA Encircling	176400	93600	mm	213.50	6.0000	mm
59	4.00	59	test_jung2	Circ	ECA Encircling	331200	43200	mm	213.00	4.0000	mm
60	4.86	236	test_jung2	Circ	ECA Encircling	18000	50400	mm	213.75	4.5000	mm
61	6.10	211	test_jung2	Circ	ECA Encircling	172800	360000	mm	253.25	70.500	mm
62	7.53	221	test_jung2	Circ	ECA Encircling	172800	28800	mm	102.50	3.0000	mm
63	8.40	37	test_jung2	Circ	ECA Encircling	172800	28800	mm	120.50	2.0000	mm
64	3.83	212	test_jung2	Circ	ECA Encircling	176400	21600	mm	139.75	1.5000	mm
65	3.82	212	test_jung2	Circ	ECA Encircling	172800	14400	mm	162.50	1.0000	mm
66	12.23	35	test_jung2	Circ	ECA Encircling	270000	93600	mm	204.25	9.5000	mm
67	12.45	215	test_jung2	Circ	ECA Encircling	93600	86400	mm	204.50	7.0000	mm
68	11.88	28	test_jung2	Circ	ECA Encircling	176400	93600	mm	213.50	6.0000	mm
69	4.00	59	test_jung2	Circ	ECA Encircling	331200	43200	mm	213.00	4.0000	mm
70	4.86	236	test_jung2	Circ	ECA Encircling	18000	50400	mm	213.75	4.5000	mm
71	6.10	211	test_jung2	Circ	ECA Encircling	172800	360000	mm	253.25	70.500	mm
72	7.53	221	test_jung2	Circ	ECA Encircling	172800	28800	mm	102.50	3.0000	mm
73	8.40	37	test_jung2	Circ	ECA Encircling	172800	28800	mm	120.50	2.0000	mm
74	3.83	212	test_jung2	Circ	ECA Encircling	176400	21600	mm	139.75	1.5000	mm
75	3.82	212	test_jung2	Circ	ECA Encircling	172800	14400	mm	162.50	1.0000	mm

2. From the buttons at the bottom of the window, click **Options.** The **Options** window appears.
3. Click the **Columns** tab (see Figure 5-19). The **Visible columns** list contains all column titles currently displayed in the report, and the **Available columns** list contains all columns titles available for reporting.
4. To add columns to the report, select a column title in the **Available columns** list and,
   - Click **Insert>>** to add the column title at the beginning of the list of columns visible in the report, *or*
   - Click **Add>>** to add the column title at the end of the list of visible of columns visible in the report, *or*
   - Click **Add All>>** to add all available columns to the list of columns visible in the report.
5. To remove columns from the report, select a column title from the **Visible columns** list and,
   - Click **<<Remove** to remove the column from the report, *or*
   - Click **<<Remove All** to remove all columns from the report.

Figure 5-19 The **Columns** tab of the **Options** window



6. To organise the order in which columns appear in the report, place the cursor over the double arrow next to the column title. The cursor changes to this:

Figure 5-20 Cursor used when moving columns in a report

Once the cursor has the shape shown in Figure 5-20, click-and-drag the column title to the place where you want it in the list. In the report, columns from left to right appear from top to bottom in the **Visible columns** list.

7. Once you are done selecting and organizing the columns that will appear in your report, click **OK.** The **Options** window closes and you return to the **Report** window where columns have been rearranged according to the order defined in the **Visible columns** list.
8. If you started configuring from a display profile, you can save this report profile with your setup by selecting **Display profiles>Saved with setup** from the contextual menu that appears when you right-click anywhere on the **Report** window.

Adding Report Entries

While performing a visual analysis of the acquired data, you might need to add entries to a report. Depending on the report and on where you are in the application, you can add report entries in one of four ways:

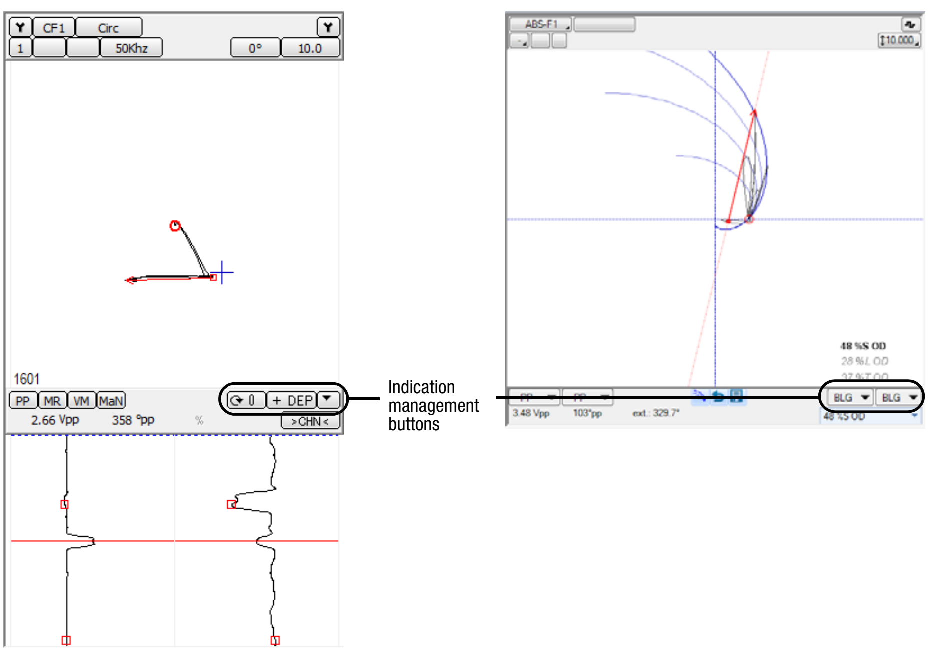
- from the Lissajous or Voltage plane window
- from the **Report** window
- from the report status
- from the Indication Codes toolbar

From the Lissajous or Voltage Plane Window

To add entries in a report from the Lissajous or voltage plane window:

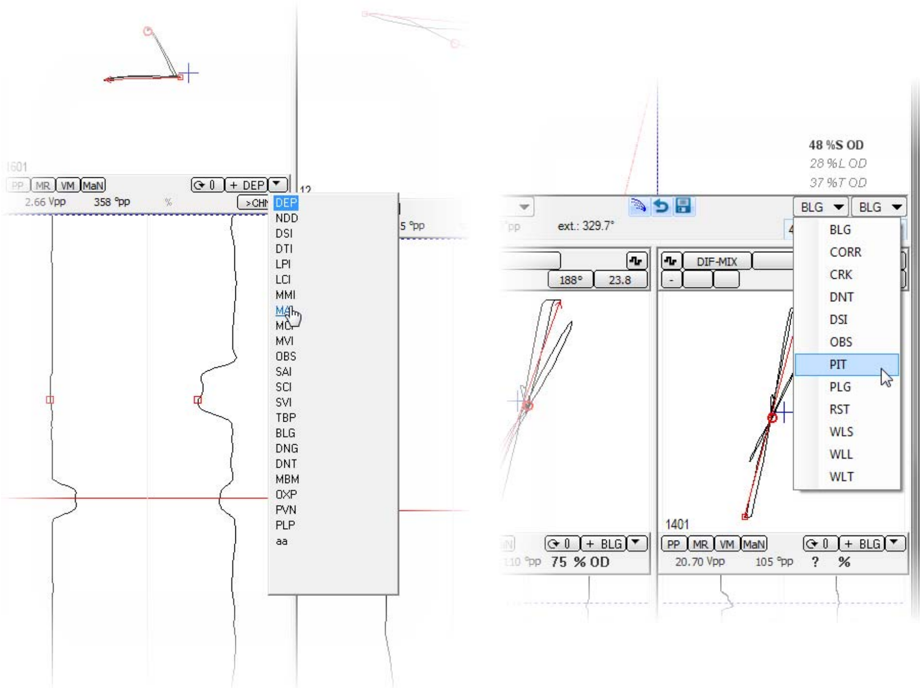
1. With a data file loaded, select **Automatic Measurements** from the **Operation** menu, or click **Automatic Lissajous measurement** from the Analysis toolbar. This activates the indication management buttons.

Figure 5-21 Active indication management buttons (Lissajous - left, voltage plane - right)



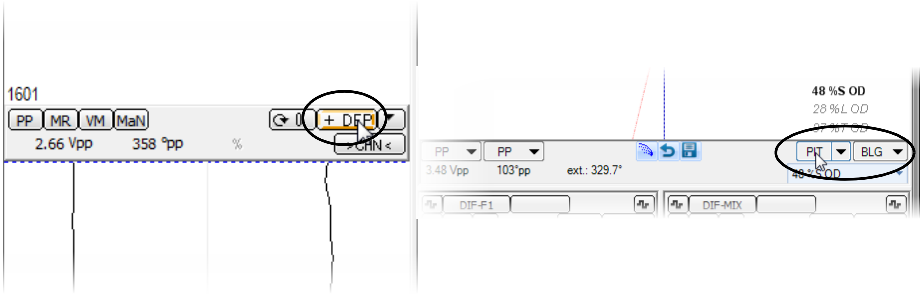
2. Click the indication selection arrow to display the type of entries available, and select the indication that you want to enter in the report. The selected indication appears in the button next to the arrow.

Figure 5-22 Available indications (Lissajous - left, voltage plane - right)



3. Click the indication button to add that type of entry to the report. The voltage, angle, and depth information displayed in the window are added to the report.

Figure 5-23 Clicking the indication button

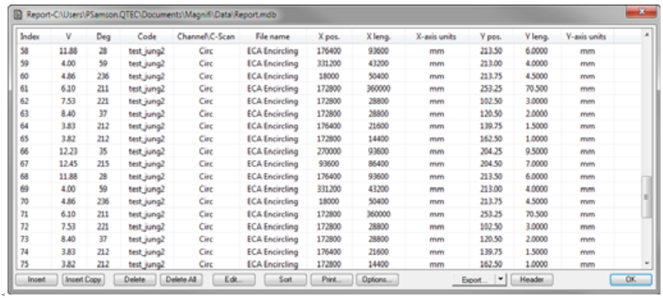


From the Report Window

To add entries in a report from the **Report** window:

1. With a data file loaded, select **Edit Report** from the **Operation** menu, or click **Edit report** from the Analysis toolbar. The **Report** window appears.

Figure 5-24 The **Report** window



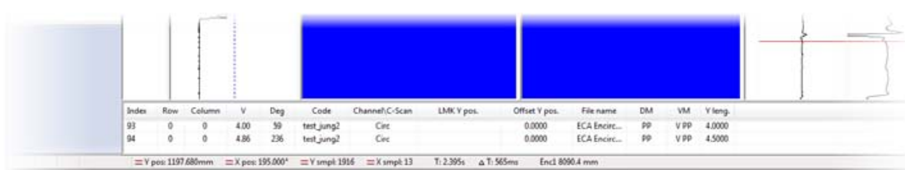
2. Click **Insert** to add a completely new report entry at the end of the list of entries, or **Insert Copy** to add a copy of the selected entry (if any) at the end of the list. An entry is added at the end of the list of entries.

From the Report Status

To add entries in a report from the report status:

1. With a data file loaded, click **Show report status** from the Analysis toolbar. The report status appears at the bottom of the data display area.

Figure 5-25 The report status



2. Right-click the report status. From the contextual menu that appears, select **Insert** to add a completely new report entry at the end of the list of entries, or **Insert Copy of Entry** to add a copy of the selected entry (if any) at the end of the list. An entry is added at the end of the list of entries.

From the Indication Codes Toolbar

The Indication codes toolbar allows you to enter *features* specified in the **Indication Codes** window by clicking on the button associated with the code. For more information on managing indication codes, see “Adding Indication Codes” on page 197.

Figure 5-26 Buttons on the **Indication codes** toolbar

Note *Clicking on the No Indication button (green check mark) adds this code to the report and automatically loads the next file.*

Deleting Report Entries

While performing a visual analysis of the acquired data, you might need to delete entries from a report. Depending on the report and on where you are in the application, you can delete report entries in one of two ways:

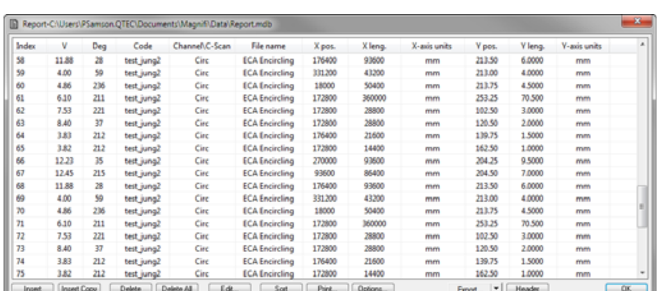
- from the Report window
- from the Report status

From the Report Window

To delete entries from the **Report** window:

1. With a data file loaded, select **Edit Report** from the **Operation** menu, or click **Edit report** from the Analysis toolbar. The **Report** window appears.

Figure 5-27 The **Report** window



2. Click **Delete** to delete the last entry in the report, or **Delete All** to delete all entries from a report.

From the Report Status

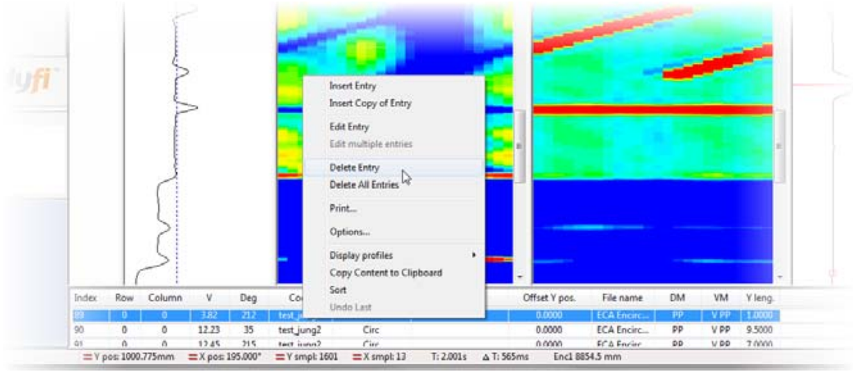
To delete entries from the report status:

1. With a data file loaded, click **Show report status** from the Analysis toolbar. The report status appears at the bottom of the data display area.

Figure 5-28 The report status

2. Right-click the report status window. From the contextual menu that appears, select **Delete** to delete the entry located under the cursor, or **Delete All Entries** to delete all entries from the report.

Figure 5-29 Deleting from the contextual menu



Note *This last step with the contextual menu can also be used with the same results in the **Report** window.*

Editing Report Entry Values

The values of all report entries (added automatically or manually) are editable. You can edit them from the **Report** window or the report status. There are many ways to edit those values:

- Directly in the **Report** window or report status (for displayed values);
- From a separate window (for all values);
- For multiple entries simultaneously.

Directly in the Report Window or Report Status

To edit report entry values directly in the report window or report status:

1. Double-click a displayed value. The value becomes editable.
2. Change the value as needed:
   - For all numerical values and most text values, type in the new value.
   - For some text values, a drop-down list appears; select a new value from it.
3. Press ENTER or click outside the text box. The modification is complete.
4. If in the **Report** window, click **OK** to close the window.

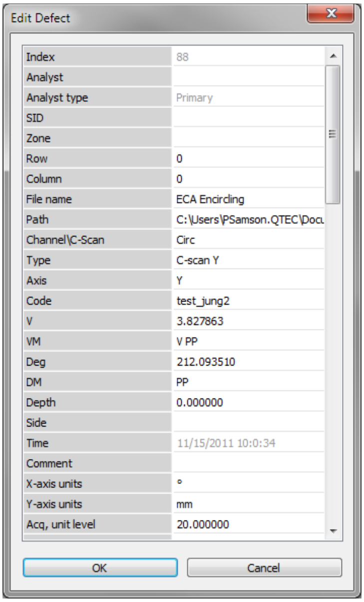
From a Separate Window

Depending on how you configured your report, all gathered data might not be displayed. However, all gathered data is still editable, but from a separate window.

To edit any of the gathered data:

1. From the **Report** window or report status, right-click a line. A contextual menu appears.
2. From the menu, select **Edit Entry.** The **Edit Defect** window appears, where all gathered data is available for edition (even data not displayed in the report).

Figure 5-30 The **Edit Defect** window



Note *From the **Report** window, clicking **Edit** after selecting a line also opens the **Edit Defect** window.*

3. If necessary, scroll in the window to find the value that you want to edit and double-click it. The value becomes editable.
4. Change the value as needed:
   - For all numerical values and most text values, type in the new value.
   - For some text values, a drop-down list appears; select a new value from it.
5. Press ENTER or click outside the text box. The modification is complete.
6. Click **OK** to close the window.

For Multiple Entries Simultaneously

If you want to apply a single value to multiple report entries, you can change all those report entries at once.

Note *This can only be done on displayed entries.*

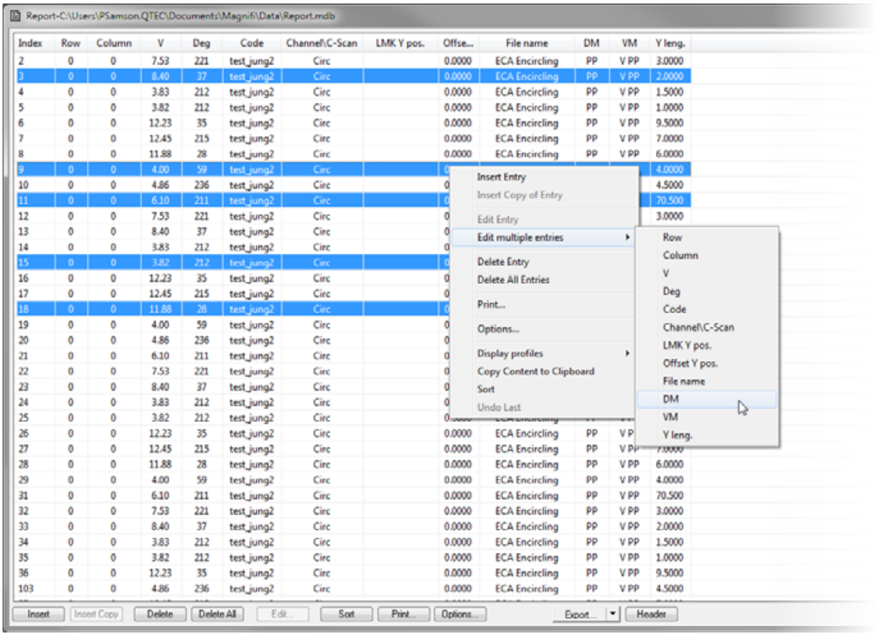
To do so:

1. From the **Report** window or the report status, select all the entries that you want to edit:

- To select contiguous entries in the list, click one entry, press the SHIFT key, and click the last entry that you want to select. All entries between the first and last selected become selected as well.
- To select non-contiguous entries in the list, press the CTRL key while clicking on all the entries that you want to select.

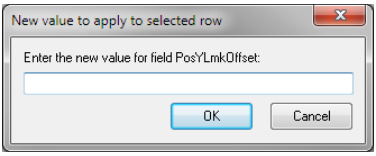
2. Right-click the last selected entry. A contextual menu appears.

Figure 5-31 Selecting multiple entries to edit



3. Select **Edit multiple entries,** and then the value that you want to edit from the sub-menu that appears (the values available in the sub-menu are the columns displayed in the **Report** window or report status). The **New value to apply to selected row** window opens.

Figure 5-32 Entering a value for multiple rows



4. Enter (either type, or select from a drop-down list, depending on the type of value to change) the value that you want to apply to all selected rows.
5. Click **OK.** The value entered is applied to all selected rows.

Viewing Data from a Report Entry

Magnifi allows you to display the specific data on which a report entry is based.

To do so:

1. Open the **Report** window.
2. Double-click the index number of the defect whose data you want to view.
   The file containing the selected defect is automatically loaded, and the cursors are positioned at the defect location.

Sorting Report Entries

You can sort the report entries if necessary.

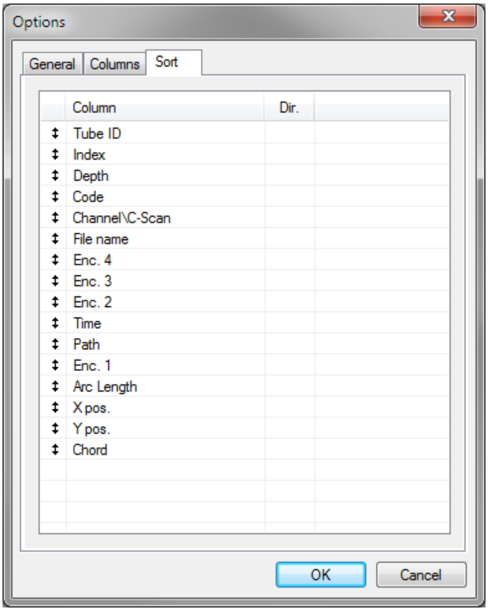
- From the **Report** window, click **Sort.** The report entries are sorted according to the defined sort order.
- From the report status, right-click anywhere in the report status, and select **Sort** from the contextual menu that appears. The entries are sorted according to the defined sort order.

Defining a Sort Order

To define a sort order:

1. From the **Report** window, click **Options** or right-click anywhere on the list (in the **Report** window or report status) and select **Options** from the contextual menu that appears. The **Options** window opens.
2. Select the **Sort** tab.

Figure 5-33 The **Sort** tab of the **Options** window

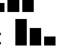


3. Organize the order in which values are sorted by placing the cursor over the double arrow next to the column title. The cursor changes to this:

Figure 5-34 Cursor used when moving values for sorting in a report

Once the cursor has the shape shown in Figure 5-34, click-and-drag the column title to the place where you want it in the list. In the example on Figure 5-33, Tube IDs are sorted first, then indication event number, indication depth, etc.

Important Eddyfi recommends sorting by file name.

4. You can choose to sort the values in each column in ascending order, descending order, or in no particular order. To select an order, double-click under **Dir.**, next to the column ID that you want to sort.
   - Double-clicking once displays the ascending order icon: 
   - Double-clicking twice displays the descending order icon: 
   - Double-clicking three times displays no icons at all, indicating that no particular sort order will be used for values in this column.
5. Once you have defined your sort order, click **OK.** The sort order is saved and will be used the next time you click **Sort** in the **Report** window, or select **Sort** from the contextual menu.

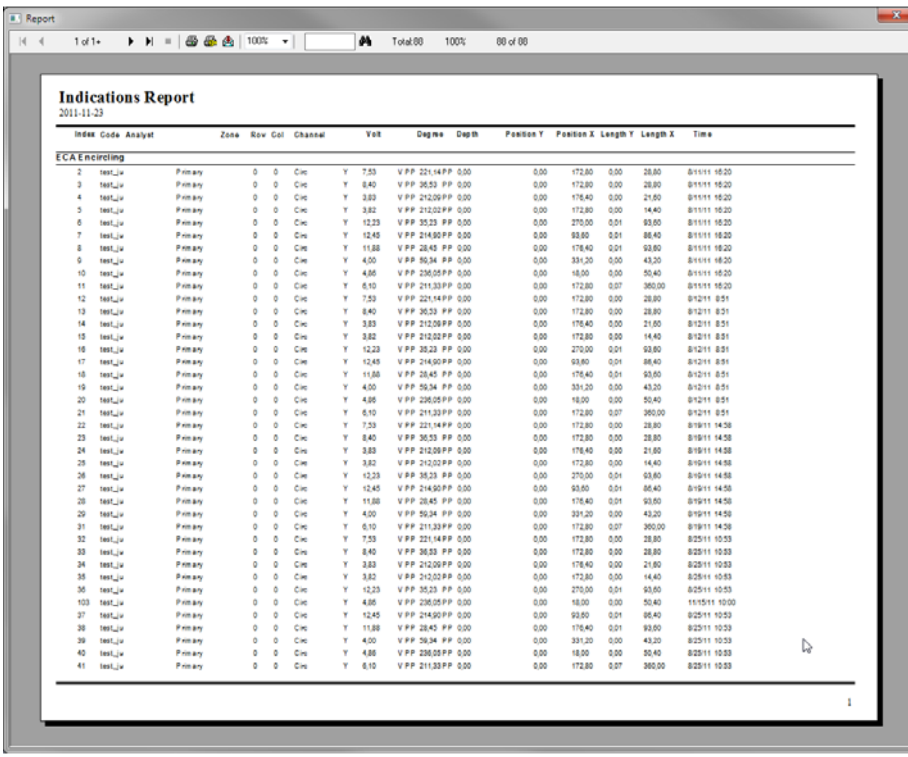
Printing/Previewing Reports

After you have finished editing your acquisition report, you can print it, if necessary.

To do so:

1. From the **Report** window, click **Print** or right-click anywhere on the list (in the **Report** window or report status) and select **Print** from the contextual menu that appears. The **Report** *preview* window opens.
   The report is formatted in a certain way that is not necessarily representative of the way data is displayed in the **Report** window itself.

Figure 5-35 The **Report** *preview* window



2. Click **Print Setup** (if necessary) to configure your printing setup. This simply opens your default printer driver configuration. Refer to your printer documentation for more information on configuring your printer driver.

3. Click **Print.** The report, as displayed in the **Report** *preview* window, is printed.

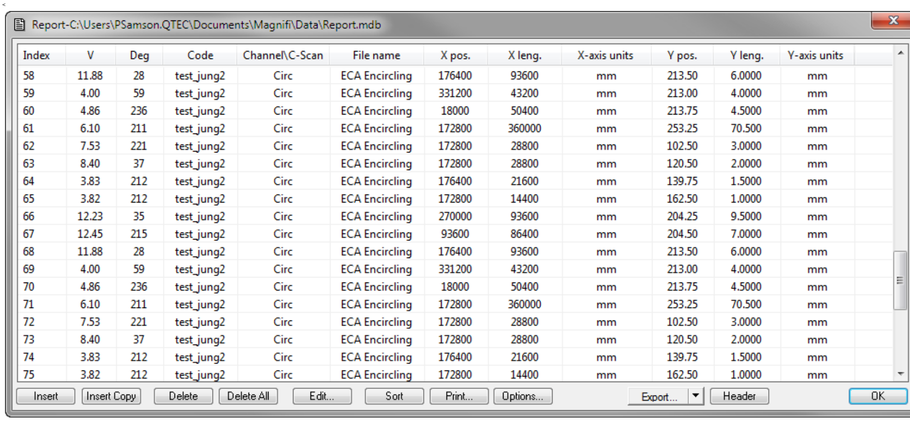
Exporting Reports

Magnifi allows you to export your reports in various common formats. Exporting is performed from the **Report** window.

To export a report:

1. Select **Edit Report** from the **Operation** menu, or click **Edit report** from the Analysis toolbar. The **Report** window appears.

Figure 5-36 The **Report** window



Index	V	Deg	Code	Channel\C-Scan	File name	X pos.	X leng.	X-axis units	Y pos.	Y leng.	Y-axis units
58	11.88	28	test_jung2	Circ	ECA Encircling	176400	93600	mm	213.50	6.0000	mm
59	4.00	59	test_jung2	Circ	ECA Encircling	331200	43200	mm	213.00	4.0000	mm
60	4.86	236	test_jung2	Circ	ECA Encircling	18000	50400	mm	213.75	4.5000	mm
61	6.10	211	test_jung2	Circ	ECA Encircling	172800	360000	mm	253.25	70.500	mm
62	7.53	221	test_jung2	Circ	ECA Encircling	172800	28800	mm	102.50	3.0000	mm
63	8.40	37	test_jung2	Circ	ECA Encircling	172800	28800	mm	120.50	2.0000	mm
64	3.83	212	test_jung2	Circ	ECA Encircling	176400	21600	mm	139.75	1.5000	mm
65	3.82	212	test_jung2	Circ	ECA Encircling	172800	14400	mm	162.50	1.0000	mm
66	12.23	35	test_jung2	Circ	ECA Encircling	270000	93600	mm	204.25	9.5000	mm
67	12.45	215	test_jung2	Circ	ECA Encircling	93600	86400	mm	204.50	7.0000	mm
68	11.88	28	test_jung2	Circ	ECA Encircling	176400	93600	mm	213.50	6.0000	mm
69	4.00	59	test_jung2	Circ	ECA Encircling	331200	43200	mm	213.00	4.0000	mm
70	4.86	236	test_jung2	Circ	ECA Encircling	18000	50400	mm	213.75	4.5000	mm
71	6.10	211	test_jung2	Circ	ECA Encircling	172800	360000	mm	253.25	70.500	mm
72	7.53	221	test_jung2	Circ	ECA Encircling	172800	28800	mm	102.50	3.0000	mm
73	8.40	37	test_jung2	Circ	ECA Encircling	172800	28800	mm	120.50	2.0000	mm
74	3.83	212	test_jung2	Circ	ECA Encircling	176400	21600	mm	139.75	1.5000	mm
75	3.82	212	test_jung2	Circ	ECA Encircling	172800	14400	mm	162.50	1.0000	mm

2. Click the down-arrow next to the **Export** button. A list of export formats appears.
3. Select the format in which you want to export. The most commonly used are:
   - **Text File:** for future import in TubePro™.
   - **Carto:** for future import in Carto™.
   - **Clipboard:** for future use in general software like Microsoft® Word® or Excel®.
4. Click **Export.** A standard **Save As** window appears.
5. Browse to the location where you want to export the report and click **Save.** The report is exported in the selected format.

Maintenance & Troubleshooting

6

Quick Troubleshooting Guide

Table 6-1 Problems — Possible Causes — Solutions

Problems	Possible Causes	Solutions
The Ectane does not appear in the Magnifi connection window.	Network card not set properly.	Make sure that your network card is in DHCP protocol. (Please refer to computer manufacturer documentation.)
	Network settings incorrect on the Ectane	Make sure that your Ectane is configured for DHCP protocol (see "Configuring Ectane for DHCP Protocol" on page 294).
	Corrupted firmware	Try and boot from the spare firmware to reprogram the Ectane (see "Fixing Corrupted Ectane Firmware" on page 292).
	Defective or unplugged network cable	Change network cable and make sure that the two network LEDs (connections and 100 Mbits) light up (please refer to your Ectane documentation for LED location).
	Internal defect on the Ectane	Call Eddyfi.
The Ectane appears in the Magnifi connection window, but a red X is displayed in the **Status** column.	Incompatible IP addresses	Make sure that the network card IP addressing scheme is identical to the Ectane's (i.e., both in DHCP or both in fixed IP address). (See "Configuring Ectane for Static IP Addresses" on page 295.)
	Ectane firmware outdated	Update Ectane firmware (see "Upgrading the Ectane Firmware" on page 290).
	Windows places the network card in power-saving mode.	Forcefully renew the card's IP address with the "ipconfig / renew".command.

Table 6-1 Problems — Possible Causes — Solutions *(continued)*

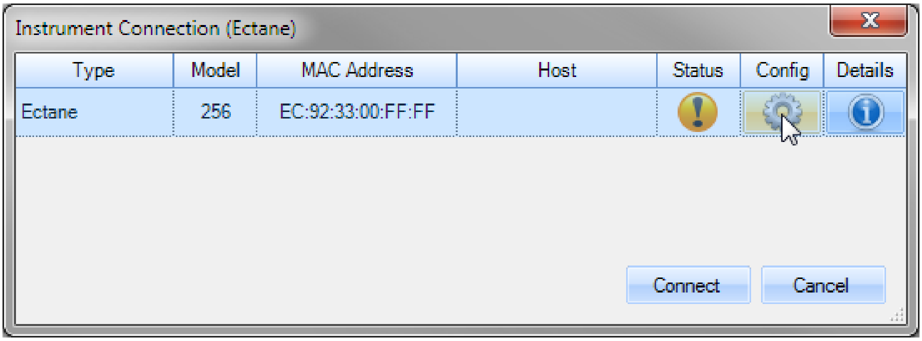
Problems	Possible Causes	Solutions
One of the network LEDs does not light up	Defective or unplugged network cable	Change network cable and make sure that the two network LEDs (connections and 100 Mbits) light up (please refer to your Ectane documentation for LED location).
	Network card settings	Make sure that settings are configured to DHCP and 100 Mbits. (Please refer to computer manufacturer documentation.)
	Internal defect on the Ectane	Call Eddyfi.
The Ectane is not charging batteries (charge indicator).	Defective or unplugged power pack	Make sure that green LED on the power pack and the Ectane (near the charge connector) lights up.
	No batteries in the Ectane	Put batteries in the Ectane.
	Internal defect on the Ectane	Call Eddyfi.
Balancing operation failed.	Probe did not have a good contact with the sample during balance.	Redo the balancing operation, ensuring that the probe is in contact during the entire process.
	The signal generated by the probe is too high.	Reduce the injection voltage and/or gain (see "Configuring Probe Topologies" on page 88).
	The probe is damaged.	Try another probe, if available.
	The operating frequency is outside the probe limits.	Make sure that you are using the probe in the appropriate frequency range.

Upgrading the Ectane Firmware

The first time that you establish communication between the Ectane and Magnifi (by clicking on **Connect to an instrument** from the toolbar, or selecting it from the **Operation** menu), you are informed of the communication status:

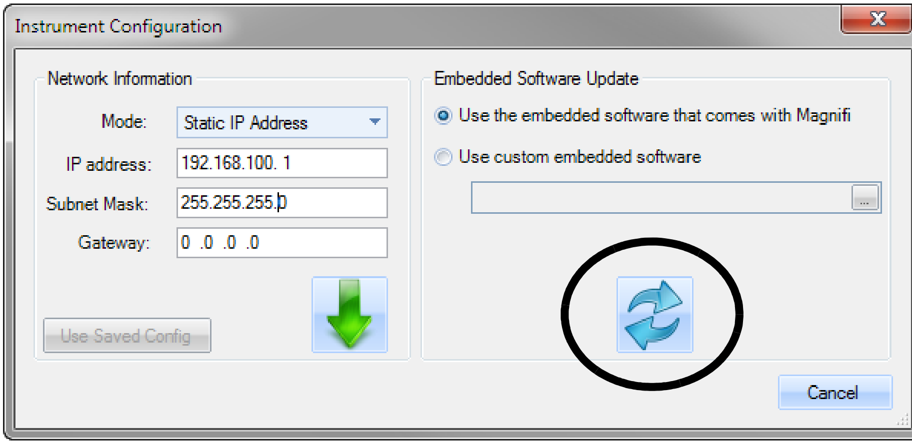
- Green check mark: communication is correct. The Ectane is running a perfectly operational firmware.
- Yellow exclamation mark: The Ectane is running a version of the firmware that is more recent than what Magnifi expected. You can still connect to the instrument and perform tasks as usual.
- Red “X” mark: The Ectane is running an older version of the firmware, not recognized by Magnifi. In these situations, you need to update the firmware as explained below.

Figure 6-1 The **Instrument Connection** window with a yellow exclamation mark



1. The window that appears is similar to the one above, but a red “X” replaces the yellow exclamation mark in the **Status** column.
2. Click **Config.** The following window appears.

Figure 6-2 Upgrading the Ectane firmware



3. Select **Use the embedded software that comes with Magnifi** from the **Embedded Software Update** section.
4. Click Send firmware to the instrument (the circling-blue-arrow button). The Ectane firmware is updated to perfectly match your version of Magnifi.
5. Once the upgrade is complete, you will have to restart both Magnifi and the Ectane.

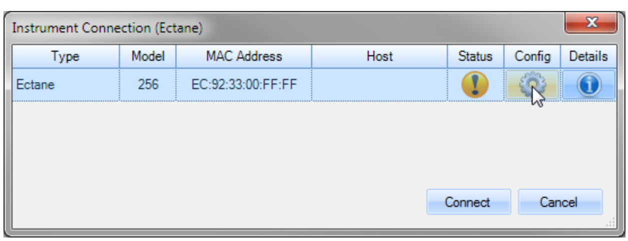
Fixing Corrupted Ectane Firmware

A sign of a corrupted firmware is that the Ectane does not appear in the **Instrument Connection** window. There are other reasons why an Ectane instrument would not appear in the **Instrument Connection** window (see Table 6-1 on page 288).

To fix a corrupted firmware:

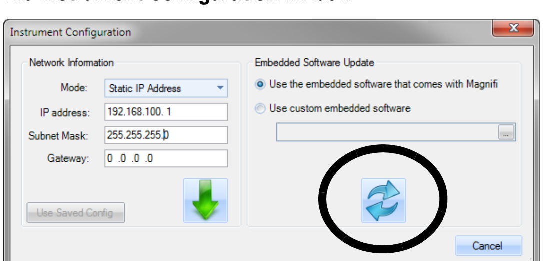
1. Open the Ectane battery compartment and locate the backup firmware button next to the battery slot (please refer to the Ectane instruction manual to locate this button).
2. Press the backup firmware button, then press the power switch.
3. Release the backup firmware button and wait for the Ectane to boot completely.
4. 
Click Connect to an instrument or select **Connect to Instrument** from the **Operation** menu. The **Instrument Connection** window appears. The Ectane should now appear in the window (if not, call Eddyfi).

Figure 6-3 The **Instrument Connection** window



5. Click in the **Config** column. The **Instrument Configuration** window appears.

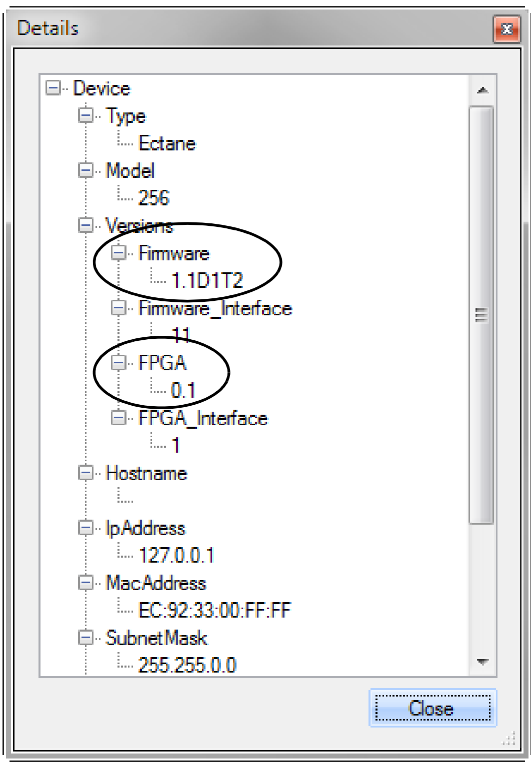
Figure 6-4 The **Instrument Configuration** window



6. Click the circling-blue-arrow button. The firmware is updated and the Ectane reboots automatically. Click OK in the window that appears. Your Ectane firmware has been fixed.

Note *It is possible that the firmware contained in your version of Magnifi is not the latest to go with your instrument. This should not prevent the Ectane from appearing in the* ***Instrument Connection*** *window, but it could cause instability problems when using the instrument. To make sure that you have the proper firmware, click* ***Details*** *in the* ***Instrument Connection*** *window. In the* ***Details*** *window that appears, make sure that the numbers indicated under* ***Firmware*** *and* ***FPGA*** *are identical to the ones indicated in your calibration certificate. If they are not, call Eddyfi.*

Figure 6-5 The **Details** window



Configuring Ectane for DHCP Protocol

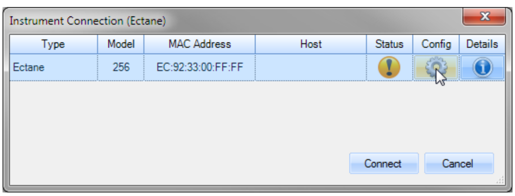
One of the advantages of the Ectane is the use of the DHCP addressing scheme.

To change the Ectane addressing scheme:

1. Launch Magnifi.

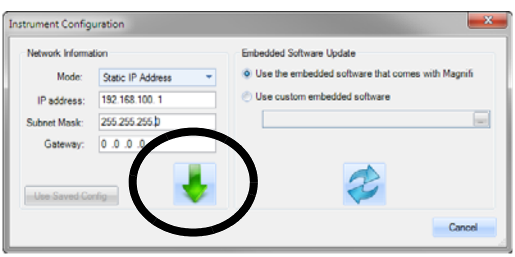
2. Click Connect to an instrument or select **Connect to Instrument** from the **Operation** menu. The **Instrument Connection** window appears.

Figure 6-6 The **Instrument Connection** window



3. Click in the **Config** column. The **Instrument Configuration** window appears.

Figure 6-7 The **Instrument Configuration** window



4. In the **Network Information** section, select **DHCP** from the **Mode** drop-down list. This deactivates all fields in the section.
5. Click the downward pointing green arrow. The Ectane is now ready to be used in DHCP mode.

Configuring Ectane for Static IP Addresses

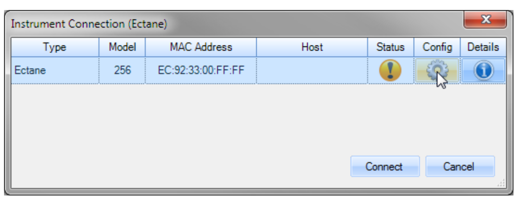
One of the advantages of the Ectane is the use of the DHCP addressing scheme, which is not compatible with older instruments (like a TC7700™ or an MS5800™). If you want to continue using these instruments and use the Ectane as well, you need to change the Ectane's hardware configuration. By switching your Ectane from a DHCP addressing scheme to a static IP address, you can continue using older instruments while still being able to work with the Ectane.

To change the Ectane addressing scheme:

1. Launch Magnifi.

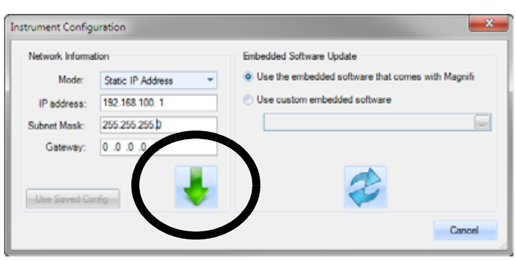
2. Click Connect to an instrument or select **Connect to Instrument** from the **Operation** menu. The **Instrument Connection** window appears.

Figure 6-8 The **Instrument Connection** window



3. Click in the **Config** column. The **Instrument Configuration** window appears.

Figure 6-9 The **Instrument Configuration** window



4. In the **Network Information** section, select **Static IP Address** from the **Mode** drop-down list. This activates all fields in the section.
5. Enter an IP address in the range recognized by your network card and click the downward pointing green arrow. It is now possible to use older instruments while still using the Ectane.

Reinstalling Magnifi

A

Reinstalling the Software

To reinstall Magnifi, simply follow the instructions from the Installation Wizard.

Note *When installation is complete, Windows 7 Program Compatibility Assistant might require that you confirm that Magnifi installed correctly (see page 301). Simply click* ***This program installed correctly.***

Figure A-1 Launching the Setup wizard

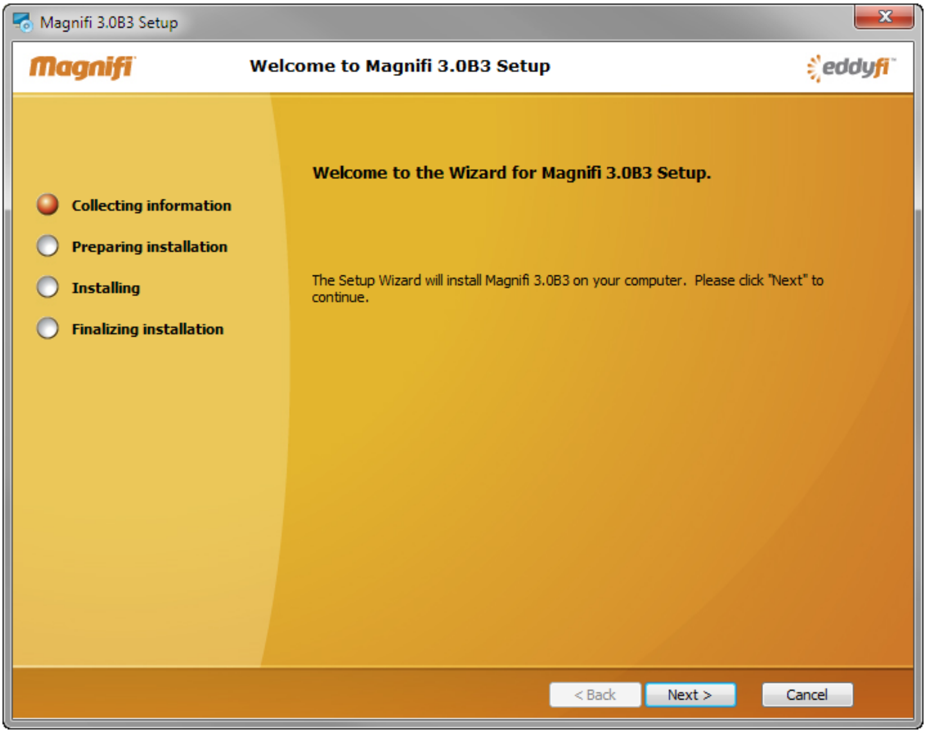


Figure A-2 Accepting the license agreement

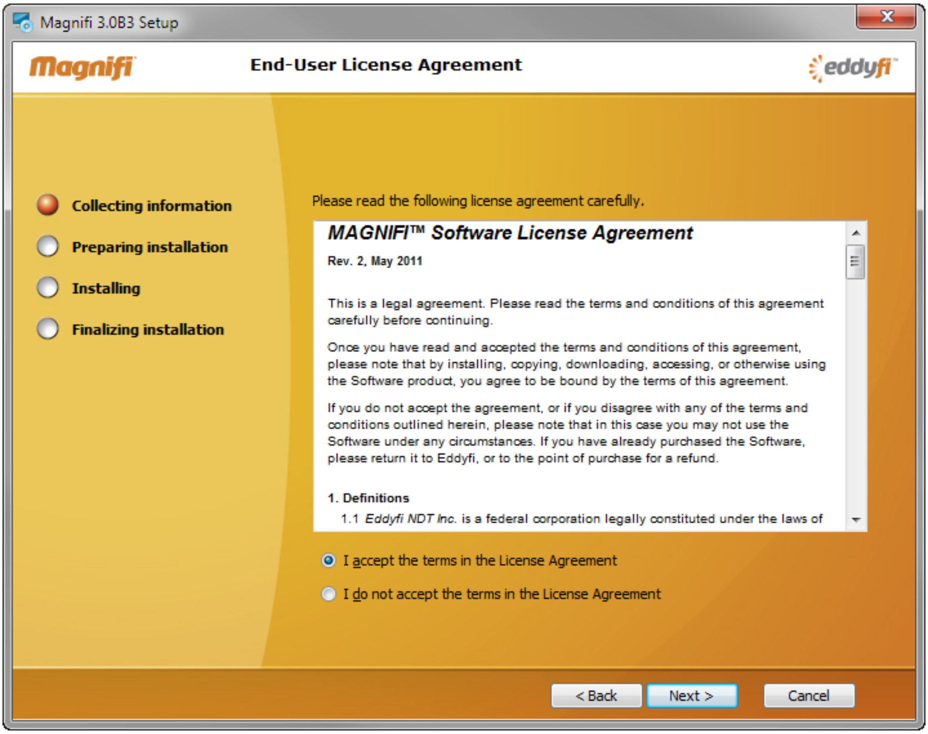
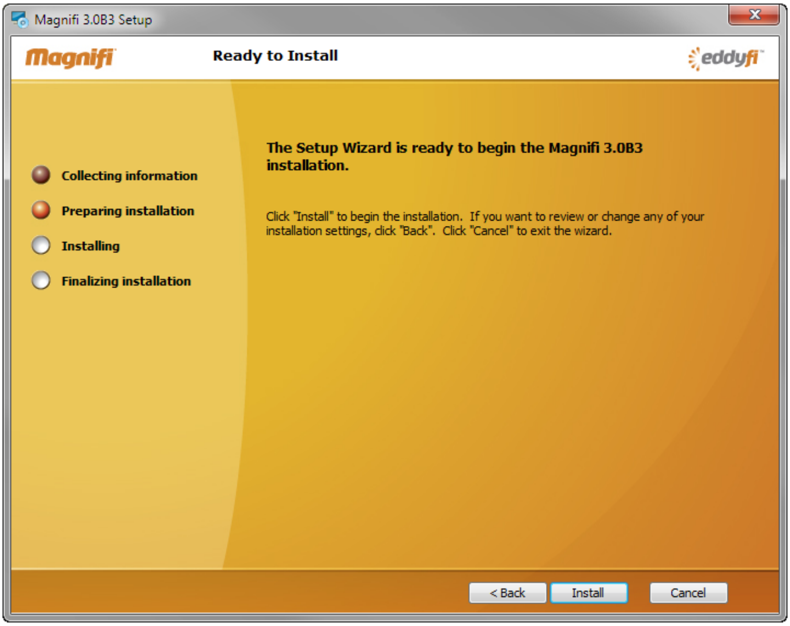


Figure A-3 Installing the software



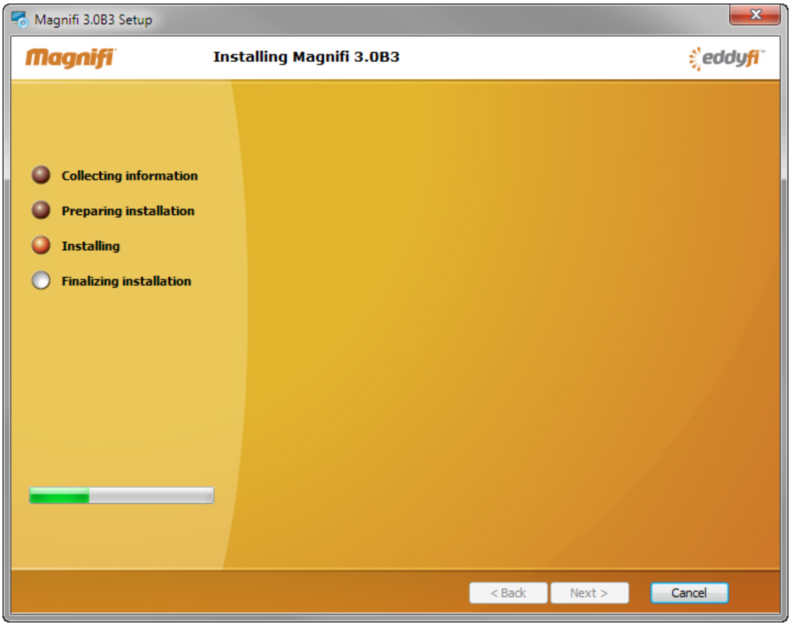


Figure A-4 Finishing the installation

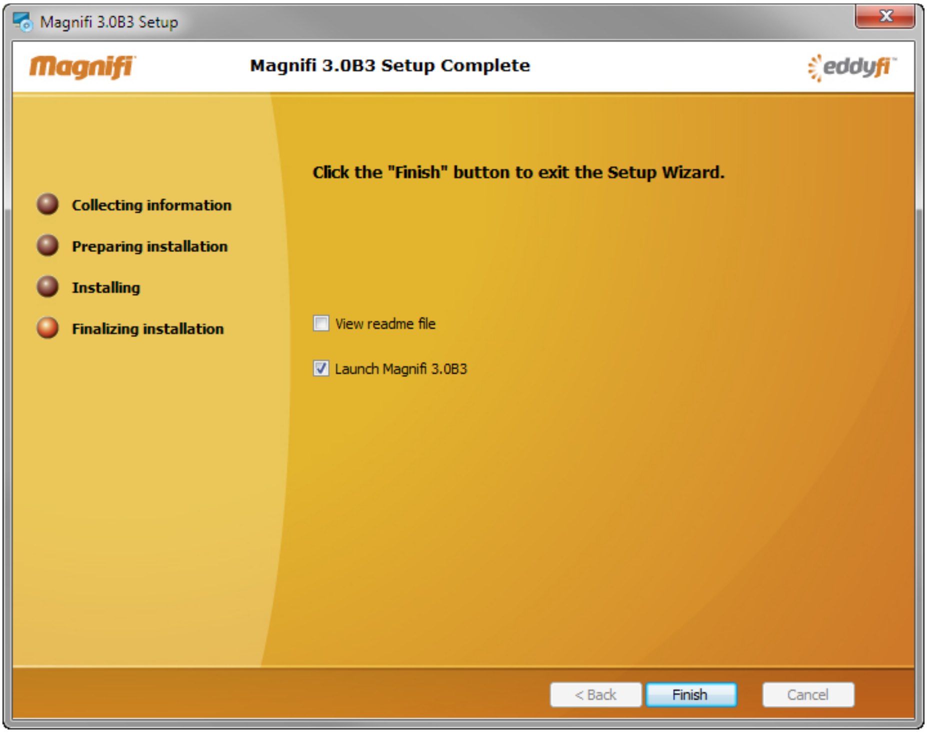
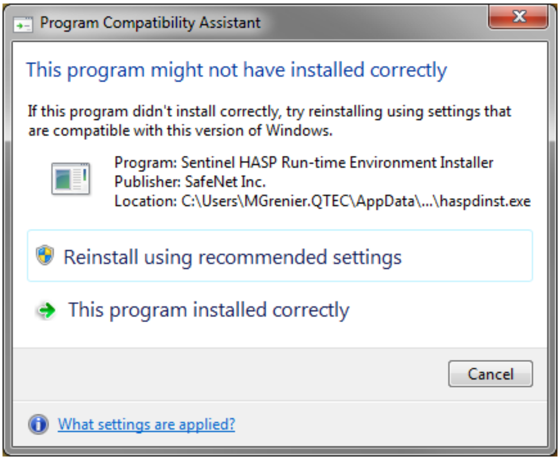


Figure A-5 Clicking "This program installed correctly"



Managing License Keys

B

Upgrading Hardware License Keys

Obtaining Upgrade Codes

To obtain the upgrade code, save the license file (*.*txt*) that was emailed to you by Eddyfi.

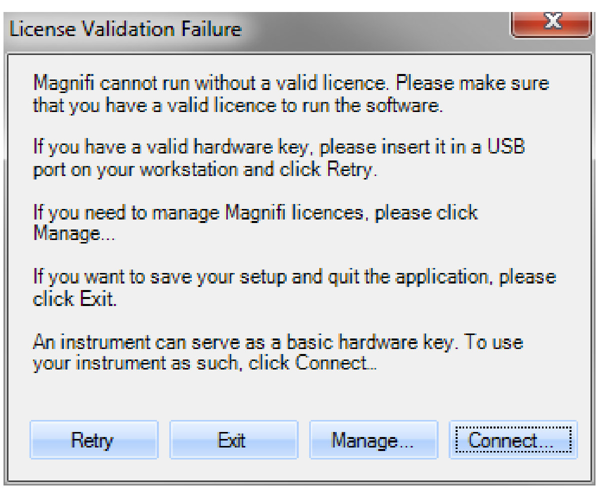
Upgrading to New License Keys

To upgrade your hardware license key:

1. Start Magnifi.
2. Select **Manage License** from the **Special** menu.

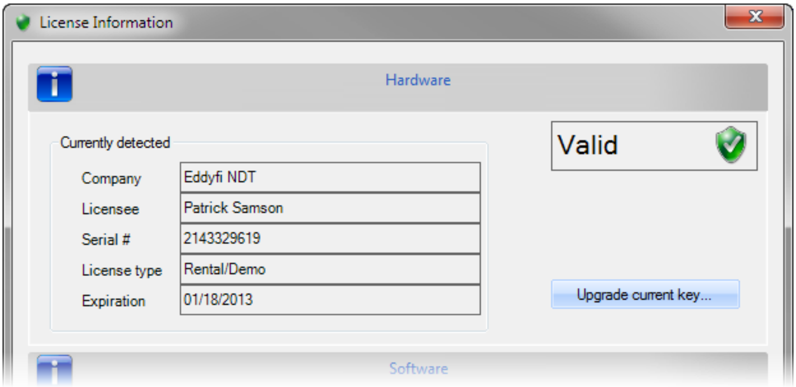
Note *If no license is installed, the **License Validation Failure** window appears.*

Figure B-1 The **License Validation Failure** window



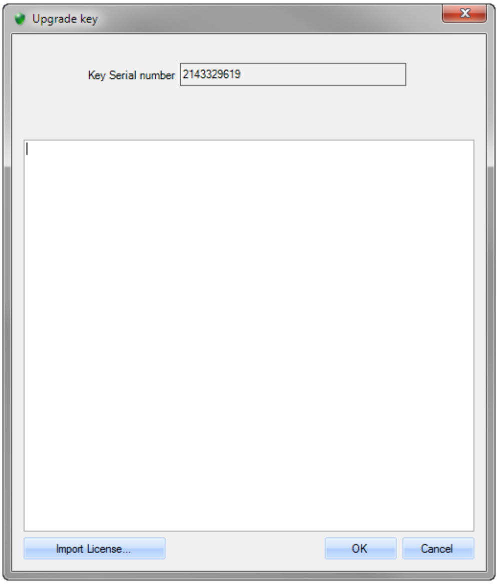
3. Click **Manage.** The **License Information** window appears.
4. Expand the **Hardware** section.

Figure B-2 The **License Information** window



5. Click **Upgrade current key.** The **Upgrade key** window opens.

Figure B-3 The **Upgrade key** window



6. Click **Import License.** A standard **Open** window appears.
7. Browse to the location of the license file (in *.txt* format), select the file and click **Open.** The license key appears in the field at the bottom of the **Upgrade key** window.
8. Click **OK.** The license key information appears in the **Currently detected** fields of the **Hardware** section. Your hardware key is upgraded.

Upgrading Software License Keys

To upgrade you software license key, you need to:

1. Release the existing license;
2. Request a new license key;
3. Upgrade to the current license.

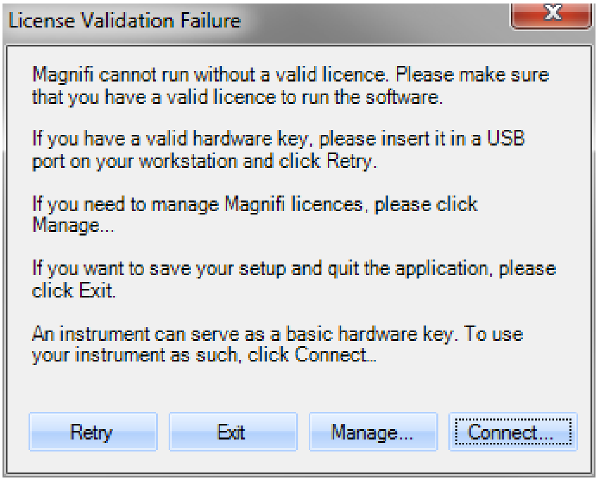
Releasing Existing License Keys

To release your existing software license key:

1. Start Magnifi.
2. Select **Manage License** from the **Special** menu.

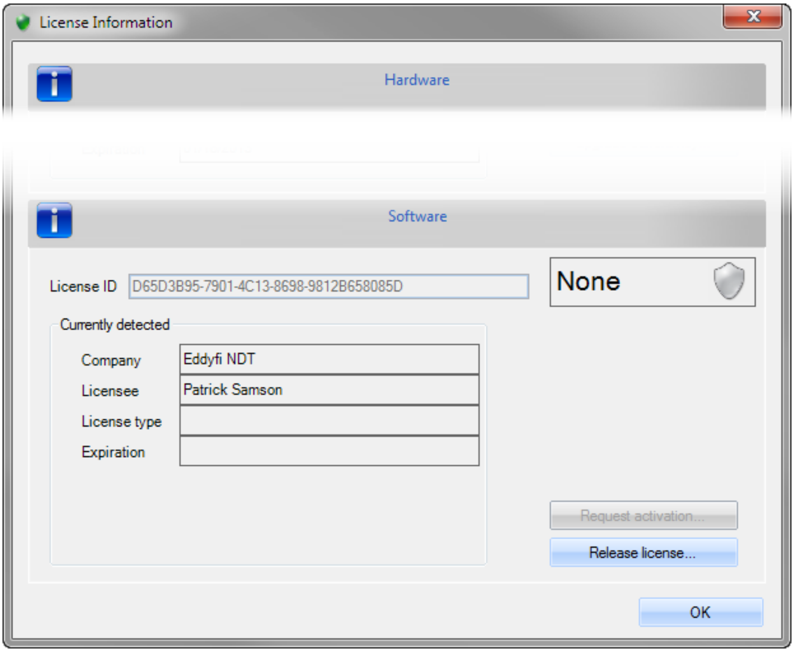
Note *If no license is installed, the* ***License Validation Failure*** *window appears.*

Figure B-4 The **License Validation Failure** window



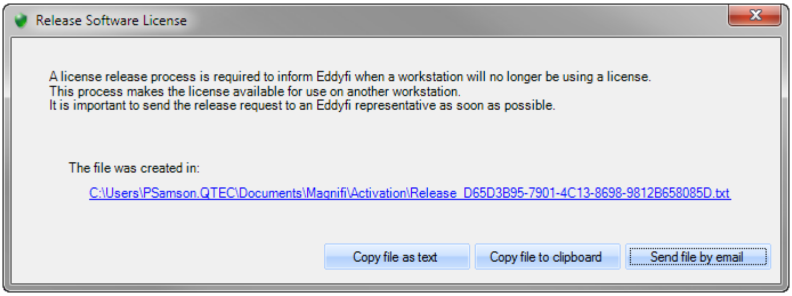
3. Click **Manage.** The **License Information** window appears.
4. Expand the **Software** section.

Figure B-5 The **Licence Information** window



5. Click **Release license.** You are asked to confirm you choice of releasing the license. The **Release Software License** window appears.

Figure B-6 The **Release Software License** window



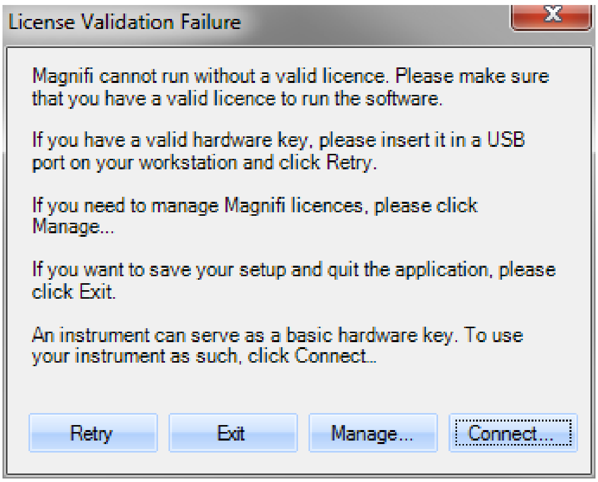
6. Click **Send file by email.** An email is automatically created and properly formatted from your default email software.
7. Send this email "as is", without changing the email's content.
8. Close the **Release Software License** window.

Requesting New License Keys

To request a new software license key:

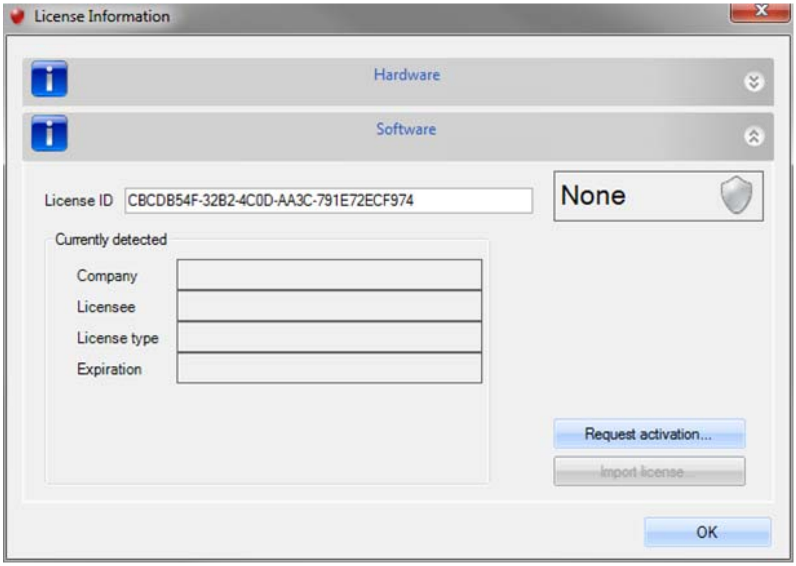
1. Start Magnifi. Since you must have released the software license prior to upgrading to a new one, the **License Validation Failure** window appears.

Figure B-7 The **License Validation Failure** window



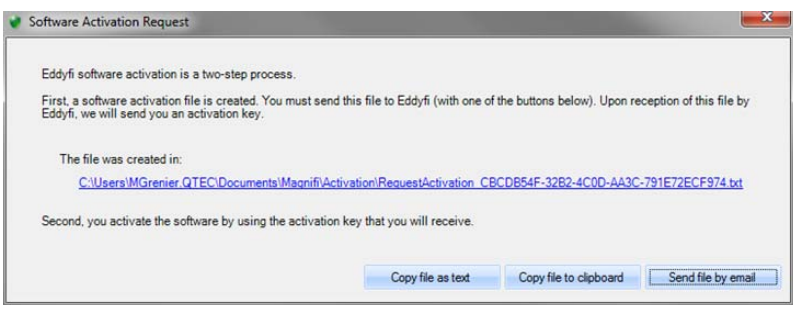
2. Click **Manage.** The **License Information** window appears.
3. Expand the **Software** section.

Figure B-8 The **License Information** window



4. Click **Request Activation.** The **Software Activation Request** window appears.

Figure B-9 The **Software Activation Request** window



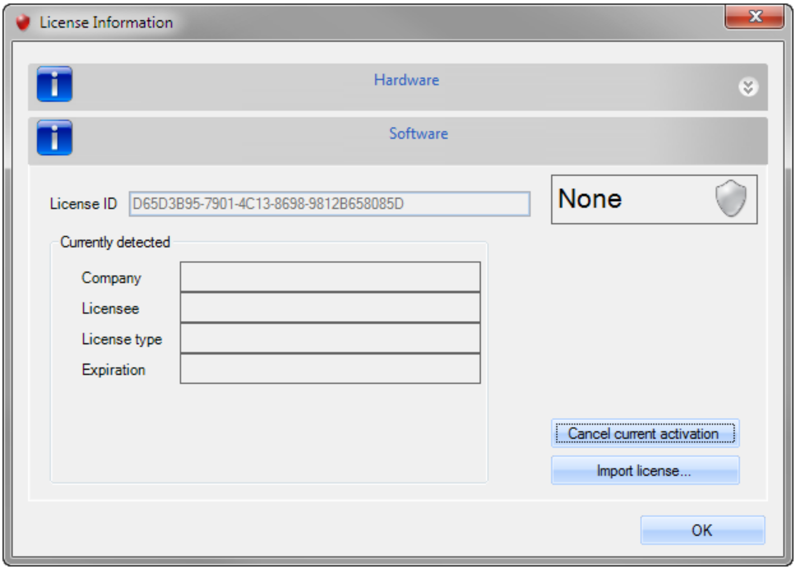
5. Click **Send file by email.** An email is automatically created and properly formatted from your default email software.
6. Send this email “as is”, without changing the email’s content.
7. Close the **Software Activation Request** window. Once your request has been validated by our purchasing department, you will receive a license file (in *.txt* format) by email.
8. Save this license file in a commonly used folder on your workstation.

Upgrading to New License Keys

To upgrade to the new license key:

1. Select **Manage License** from the **Special** menu. The **License Information** window appears.
2. Expand the **Software** section.

Figure B-10 The **License Information** window



3. Click **Import license.** A standard **Open** window appears.
4. Browse to the location of the license file (in *.txt* format), select it and click **Open.** The license key information appears in the **Currently detected** fields of the **Software** section. Your software key is upgraded.

Operating TC7700 & MS5800

C

Installing Software for the TC7700 and MS5800

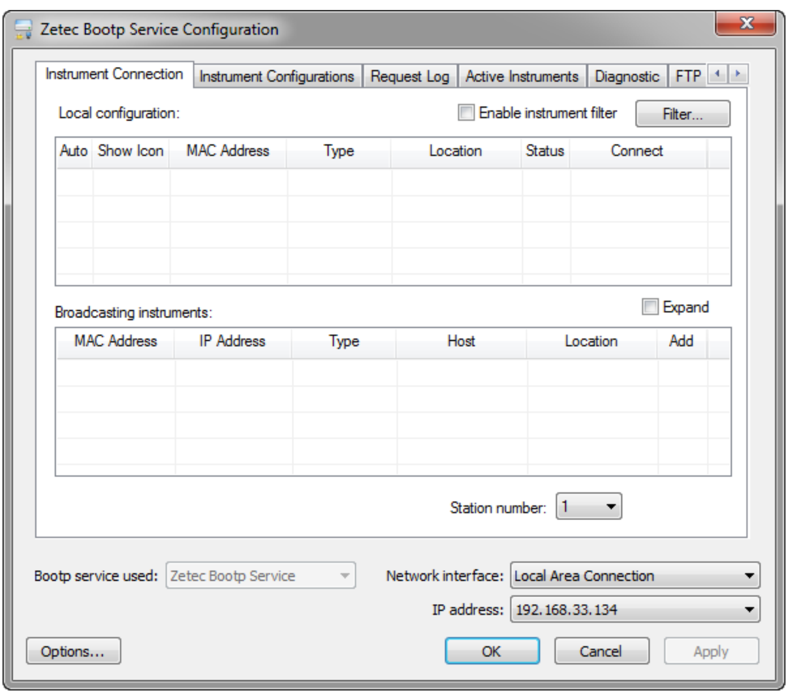
Magnifi comes with a separate installer for all the software necessary to establish communication and ensure compatibility with TC7700™ and MS5800™ test units. For the sake of brevity, we will call these test units “legacy” in the following pages.

To install this software:

1. On the installer disk that came with Magnifi, browse and find the *MS5800 and TC7700 support for Magnifi 3.3.exe* file.
2. Double-click the file.
3. Follow the instructions on the screen to complete the installation.

For details about how to properly use the Bootp software to establish the first line of communication between your workstation and the test unit, refer to the Zetec Bootp technical documentation.

Figure C-1 The **Zetec Bootp Service Configuration** window



Establishing Communication with Magnifi

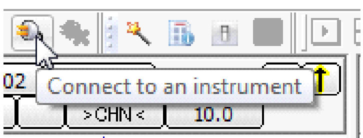
Once proper communication is established between your workstation and the test unit, establishing communication with Magnifi with the TC7700 and MS5800 is as simple as connecting with an Ectane.

Note *If you intend to use the Ectane along a TC7700 or an MS5800 with the same computer, you must configure a static IP address to communicate with the Ectane. See “Configuring Ectane for Static IP Addresses” on page 295 for details.*

To establish communication with the a TC7700 or MS5800:

1. Start the instrument.
2. Start Magnifi.
3. Load a setup made for the TC7700 or the MS5800.
4. Click **Connect to an instrument**.

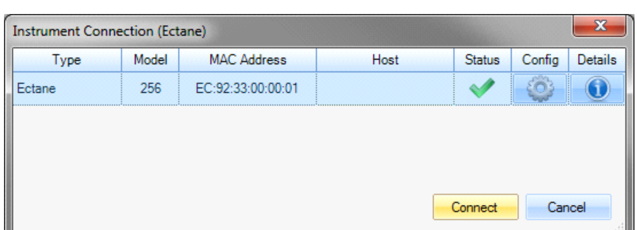
Figure C-2 Connecting to an Instrument



Note *If there are no instruments listed in the **Instrument Connection** dialog box, verify that you have an active link with instruments in the Bootp service. Also make sure that your current setup is compatible with the TC7700 or the MS5800. If your current setup is for the Ectane, Magnifi only searches for an Ectane. To load a default TC7700 setup, on the **File** menu, point to **New**, and thenclick **TC7700 Setup**.*

5. Select the available instrument, and then click **Connect**.

Figure C-3 Selecting an instrument



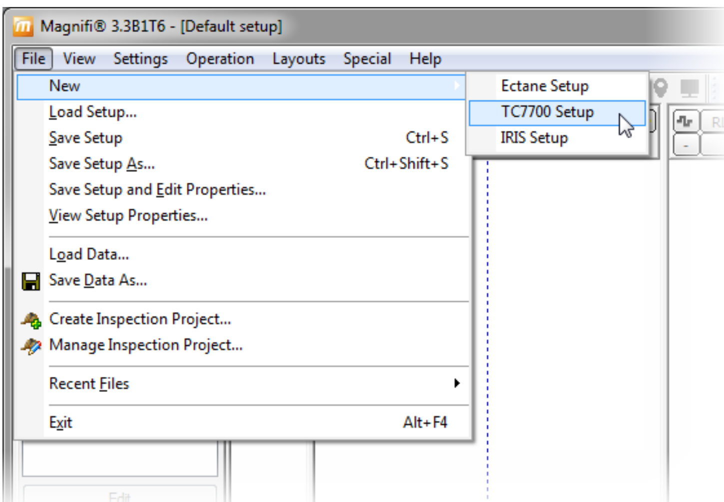
Creating a Setup for the Test Unit

Magnifi can create setup files for legacy test units. TC7700 and MS5800 use the same setup files.

To do so:

1. On the **File** menu, point to **New**, and then click **TC7700 Setup.**
   A dialog box appears to inform you that the setup in use was modified and whether you want to save it.

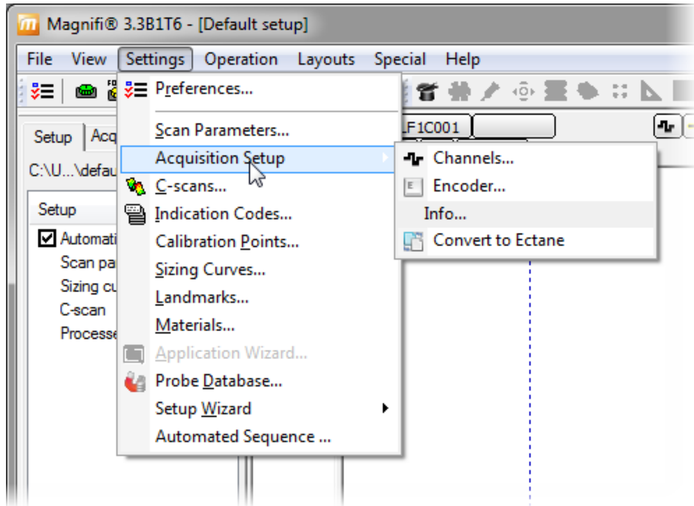
Figure C-4 Creating a new legacy unit setup file



2. To save the current setup, click **Yes.** Otherwise, click **No.**
   For more information about saving setups, see “Saving Setups” on page 212.

Note *Once the new setup file is created, a new toolbar appears and the elements of the* ***Settings>Acquisition Setup*** *menu change.*

Figure C-5 New toolbar and **Acquisition Setup** menu



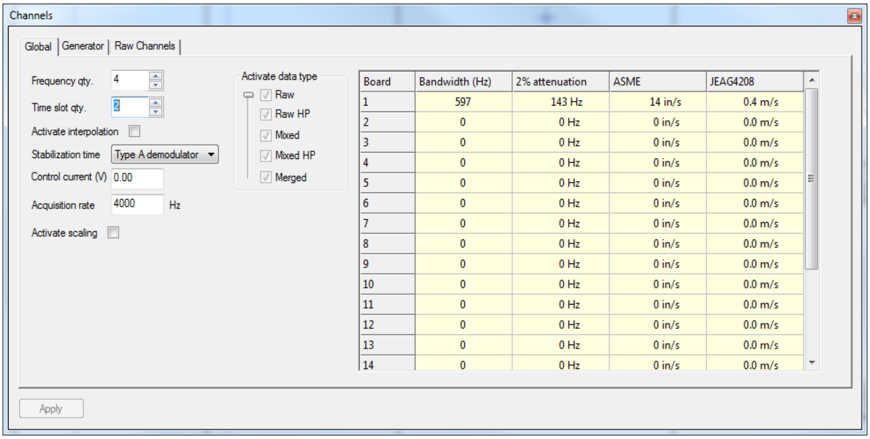
Setting Up Channels

When working with a legacy test unit like a TC7700 or MS5800, you can set various channel parameters, as explained in the following pages.

Once all three tabs are properly filled out, click **Apply.** All values are input into the TC7700/MS5800 setup.

- To access channel parameters, on the **Settings** menu, point to **Acquisition Setup**, and then click **Channels.**
  You can also click the Channels button in the TC7700 Acquisition Setup toolbar. The **Channels** dialog box appears.

Figure C-6 The TC7700/MS5800 **Channels** dialog box — **Global** tab



Defining Global Channel Parameters

As its name implies, global channel parameters affect all the channel parameters when working with a legacy unit (see Figure C-6).

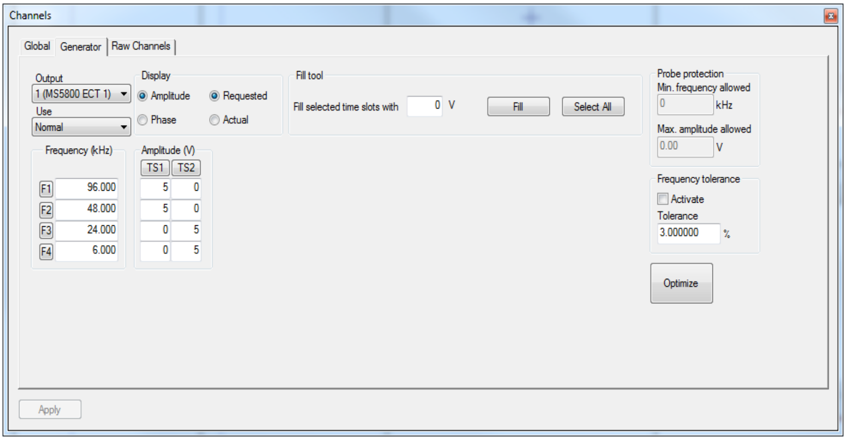
To set global channel parameters:

1. In the **Frequency qty** box, type the number of frequencies (max. 8) used for the current configuration.
2. In the **Time slot qty** box, type the number of time slots used in the current configuration. *You cannot have more than 16 time slots.*
3. To interpolate values between time slots, select the **Activate interpolation** check box.
4. When using a multiplexer, select your type of demodulator on the **Stabilization time** list. If your demodulator requires a long stabilization period, select **Type A demodulator**. Otherwise, select **Type B demodulator**.
5. In the **Control current (V)** box, type the maximum voltage for the digital-analog converter output (between 1 and 10).
6. In the **Acquisition rate** box, type the acquisition rate in Hz.
7. To scale the acquired data, select the **Activate scaling** check box.
8. You can activate gathering certain types of data by the test unit. To do so, move the slider of the **Activate data type** section. The data gathered and its associated results appear in the table on the right.

Defining Generator Parameters

As its name implies, the **Generator** tab of the **Channels** dialog box manages all the EC generator board parameters (see Figure C-7).

Figure C-7 The TC7700/MS5800 **Channels** dialog box — **Generator** tab



To configure the parameters:

1. On the **Output** list, select the generator output that you want to display or modify.
   You can only configure one generator at a time. You can have no more than four generators per generator board, and a maximum of two boards per test unit. For the MS5800, generator outputs 1 and 2 are dedicated to ECT and generator outputs 3 and 4 are dedicated to RFT.
2. On the **Use** list, assign a use to the output that you selected above.
   To enable the generator board, select **Normal**. To balance an absolute impedance probe in a setup using an internal balancing source, select **Balancing**. Unused outputs must be **Off.**
3. In the **Frequency (kHz)** section, type a value for each frequency that you want to use.
   For systems with multiplexed frequencies (simultaneous injection), the actual frequency used may vary slightly from what is requested because the system may need to alter the frequency slightly to avoid noise-inducing harmonics.
4. In the **Display** section, select whether to display and configure signals sent to the probe in volts (**Amplitude**) or in degrees (**Phase**). Also, select whether you want to display the requested (**Requested**) or real (**Actual**) value sent to the probe by the system.

Note *The name of the **Phase (deg)** section in Figure C-7 depends on your selection at the previous step.*

5. In the **Phase (deg)/Amplitude (V)** section, click the row (**F1, F2,...** in the **Frequency (kHz)** section) and column (**TS1, TS2,...**) headers to select the frequencies and time slots to which you want to apply a specific value. You can also quickly select all the time slots and frequencies by clicking **Select All** in the **Fill tool** section.
6. In the **Fill selected time slots with** text box of the **Fill tool** section, type the value that you want to send to the probe.
7. Click **Fill**.
   This sets up the amplitude or phase sent to the boxes selected at step 5.

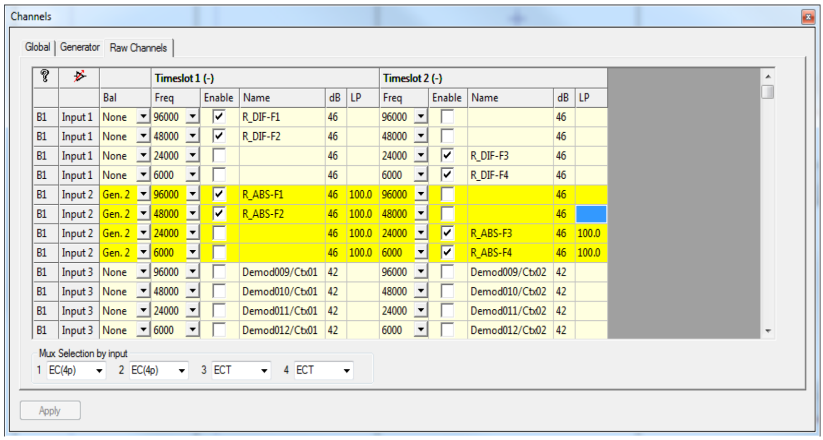
Important The maximum total value for one column cannot exceed 10 V. However, any combination of values is allowed as long as it remains within the 10 V limit.

8. If necessary to make sure that indications are within the requested percentage of tolerance, activate **Frequency tolerance**.
9. If necessary to automatically optimize the phase and amplitude to the maximum injection frequency, click **Optimize**.

Defining Raw Channel Parameters

The **Raw Channels** tab (see Figure C-8) allows you to specify the configuration of each raw (unprocessed) channel.

Figure C-8 The TC7700/MS5800 **Channels** dialog box — **Raw Channels** tab



To do so:

1. If necessary, click the "+" or "–" next to the appropriate **Timeslot** to expand or collapse the view of the timeslot.
2. For balance purposes, in the **Bal** column, click the drop-down list to select a generator to associate with the input.
3. Choose frequencies from the drop-down lists under the **Freq** column. You cannot use more that four frequencies per board.
   This is only possible if you have entered a number greater than one in the **Frequency qty** field of the **Global** tab.
4. You can enable specific inputs for each timeslot by checking the appropriate box in the **Enable** column.
5. Enter a timeslot name by clicking in the appropriate **Name** field.
6. Enter a gain value by clicking in the appropriate **dB** field.
7. Enter a low-pass filter value by clicking in the appropriate **LP** field.
8. Select the appropriate **Mux Selection by input**.
   When you use an MS5800, select the **ECT** or **EC(4p)** option if your eddy current probe is connected to the 41-pin or 4-pin connectors, respectively. Select the **RFT** or **MFL** option according to the type of probe you are using.
9. When you've completed specifying all the necessary information on the three tabs, click **Apply.**

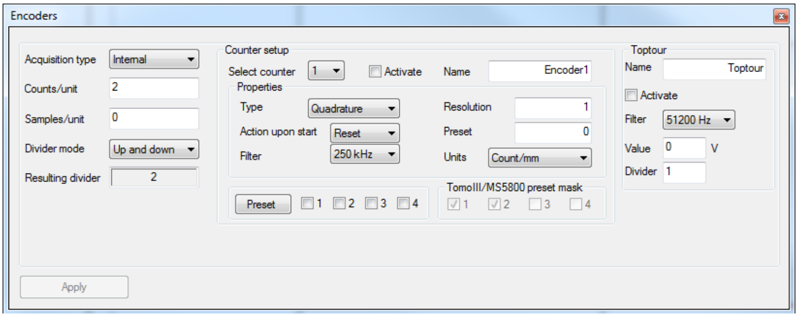
   All values are input in the TC7700 or MS5800 setup.

Setting Up Encoders

When working with a legacy test unit like an MS5800 or TC7700, you can set various encoder parameters, as explained in the following pages.

To access the encoder parameters, select **Settings>Acquisition Setup>Encoders.** You can also click the Encoder button in the TC7700 Acquisition Setup toolbar. The **Encoders** dialog box appears.

Figure C-9 The TC7700/MS5800 **Encoders** dialog box



1. When you've configured all the parameters, click Apply to sent the configuration to your instrument.
2. You can then, close the dialog box by clicking the close button in the top right corner.

Defining General Encoder Parameters

The first part of setting up encoders when working in a TC7700 is defining the general encoder parameters.

To do so:

1. In the **Encoders** dialog box, select the synchronization type to trigger the acquisition in the **Acquisition type** list.
   - **Internal**: the acquisition is triggered at the rate specified on the **Global** tab of the **Channels** dialog box (see page 316).
   - **External**: the acquisition is triggered by the rate of the external clock.
   - **Counter *n***: the acquisition is triggered by the encoder motion (the master board supports up to four encoders).
2. In the **Counts/unit** text box, type the number of counts per unit of your encoder (resolution).
3. In the **Samples/unit** text box, type the number of samples that you want per encoder count.

4. On the **Divider mode** list, select whether the acquisition is triggered when the encoder detects the signal's falling edge (**Down**), raising edge (**Up**), or either of them (**Up and down**).

Once all these parameters are defined, a number appears in the **Resulting divider** box. This is the number of encoder counts that the software uses to perform the acquisition.

Setting Up Encoder Parameters

Once the general parameters are defined, you can specify more precise encoder (counter) parameters.

To do so:

1. On the **Select counter** list, select the encoder (counter) that you want to configure.
2. To activate the encoder channel in Magnifi, select the **Activate** check box.
3. In the **Name** text box, type the encoder channel name that is created when you activate the corresponding encoder.
4. Configure the properties of the selected encoder:
   - a On the **Type** list, select the type of your encoder: **Clock direction, Quadrature, Up,** or **Down.**
   - b On the **Action upon start** list, select the action to be carried out at the start of an acquisition: **None** (no action upon start), **Reset** (the encoder is reset upon start), or **Preset** (the value in the **Preset** box [see below] is used upon start).
   - c On the **Filter** list, select the encoder low-pass filter frequency.

Note *This feature is used to filter out noise that may be present in the encoder data. Most of the time, this feature is not required. Since there is no "off" state for this feature, select the highest available value* ***(1 MHz)*** *if you want to make sure that counter data is not affected.*

   - d In the **Resolution** text box, type the encoder resolution. This information is usually found in the encoder specifications.
   - e In the **Preset** text box, type the value, in steps, that is used to preset the encoder (see step b above).
   - f On the **Units** list, select the unit type corresponding to the encoder resolution value that you typed in the **Resolution** text box.

5. Next to the **Preset** button, select the check box next to the number corresponding to the encoder that you want to preset.
6. Click **Preset** to set the selected encoder to the value entered in the **Preset** text box (see step 4e above).

Defining Toptour Channel Parameters

Once the encoder parameters are defined, you can specify more precise parameters for the toptour channel.

To do so:

1. In the **Name** text box, enter the name of the toptour channel that will be created.
2. Check the **Activate** box to activate the newly created toptour channel.
3. In the **Filter** drop-down list, select the toptour channel low-pass filter frequency used for signal debouncing.
4. In the **Value** text box, enter the amount of voltage to be sent to the probe.
5. In the **Divider** text box, enter the number of signals to be sent from the probe on each rotation.

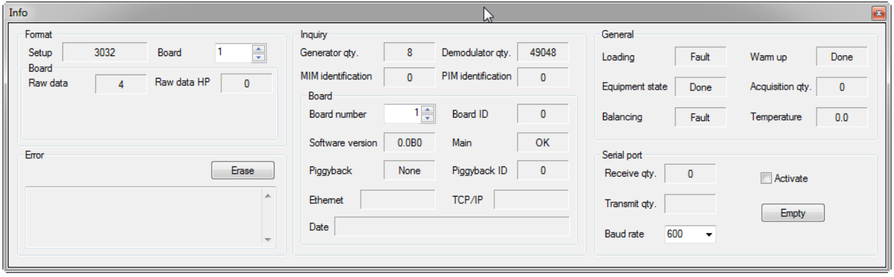
Once all these parameters are set, close the dialog box by clicking the close button in the top right corner.

Viewing Legacy Setup Info

To view setup information for your legacy unit, select **Settings>Acquisition Setup>Info.**

The **Info** dialog box appears.

Figure C-10 The TC7700/MS5800 **Info** window



In that dialog box, the following information is displayed:

1. In the **Format** sub-section:
   - **a** The **Setup** info box indicates the number of setups sent to the test unit since the last startup.
   - **b** In the **Board** text box, enter the number of the board for which you want information.
   - **c** In the **Board** group box: **Raw data** and **Raw data HP** info boxes indicate the number of bytes used per channel.

2. In the **Error** sub-section, the info box presents error codes generated by the test unit. Click **Erase** to remove all displayed errors.
3. In the **Inquiry** sub-section:
   a The **Generator qty** info box indicates the number of generators found in the test unit.
   b The **MIM identification** info box indicates the MIM (multipurpose interface module) found in the test unit.
   c The **Demodulator qty** info box indicates the number of demodulators found in the test unit.
   d The **PIM identification** info box indicates the PIM (probe interface module) found in the test unit.
   e In the **Board** group box, the **Board number** text box allows you to select the number of the board for which you need information. The **Software version** info box indicates the version of the software inside the acquisition board. The **Piggyback** info box indicates if a piggyback board is there or not. The **Ethernet** info box indicates the acquisition board Ethernet address. The **Date** info box indicates the creation date of the software version inside the acquisition board. The **Board ID** info box indicates the serial number of the selected acquisition board. The **Main** info box indicates the state of the master EC-Acquisition board. The **Piggyback ID** info box indicates the serial number of the piggyback board. The **TCP/IP** info box indicates the IP address of the acquisition board.
4. In the **General** sub-section:
   a The **Loading** info box indicates the state of the test unit loading sequence and if there were anomalies once loading is finished.
   b The **Equipment state** info box indicates the type of operation currently handled by the test unit: **INIT** (initialization), **NORMAL, CFG** (configuration), **BALANCE, CAL** (calibration), **DIAG** (diagnostic), **ACQ** (acquisition).
   c The **Balancing** info box indicates the state of the test unit balancing sequence and if there were anomalies once balancing is done.
   d The **Warm up** info box indicates the test unit warm up sequence: RUNNING and DONE.
   e The **Acquisition qty** info box indicates the number of acquisitions accomplished since the test unit or Magnifi was started (the most recent of either event).
   f The **Temperature** info box indicates the temperature of the test unit, in degrees Celsius.
5. In the **Serial Port** sub-section:
   a The **Receive qty** info box indicates the number of communications received since either the test unit or Magnifi was started (the most recent of either event).

b The **Transmit qty** info box indicates the number of communications transmitted since the test unit or Magnifi was started (the most recent of either event).

c The **Baud rate** drop-down list allows you to select a new baud rate speed.

d The **Activate** check box validates the requested modifications for the **Serial port** sub-section.

e The **Empty** button resets the data of the **Serial port** sub-section.

Mouse and Keyboard Functions

D

In Strip Charts

Table D-1 Mouse and keyboard functions

Area	Action	Pointer	Function
(Channel list)	Left click		Displays a list of channels. Click any to select.
Channel name	Left click		Changes to previous channel in list.
Channel name	Right click		Changes to next channel in list.
	Left click		Toggles the view between horizontal, vertical, amplitude, and phase components.
(Data type)	Left click		Displays a list to choose the data type.
Scale value	Left click		Changes to the previous scale on the list.
Scale value	Right click		Changes to the next scale on the list.
Scale value	Middle click		Opens a list of voltage values. Click any to select.
Scale value	Left click + vertical drag		Increases/Decreases the voltage value.
Signal area	Right click + vertical drag		Marks in green the selected area. Release the mouse button to zoom in.
Signal area	Left click		Moves the red cursors to the location of the click.
Signal area	Right click		Opens a contextual menu.
Signal area	SHIFT + left click + horizontal drag		Moves the data on the strip chart, left or right.
Measurement cursor in signal area	Left click + drag		Moves cursor.
Left side of signal area	Left click		Moves red cursor to that location, preserving spacing between cursors.
Left side of signal area	Right click		Moves the closest dotted red cursors to that location, changing the spacing between cursors.

Table D-1 Mouse and keyboard functions *(continued)*

Area	Action	Pointer	Function
Anywhere	CTRL + left click + dragging the pointer out of the strip chart		Warns that the pointer movement is invalid. Move the pointer to another place, or release the mouse button to cancel the current action.
Red dot cursors	Left click + drag		Increases or decreases the area covered inside the cursors.
100 (Frequency)	Left click / Right click		Changes to previous/next frequency for the same coil.
1 (Coil)	Left click / Right click		Changes to previous/next coil in the same frequency.
- (Link views)	Left click		Changes to previous value. ("-": Not linked, 1-9: Linked, "H": Hyperlink)
- (Link views)	Right click		Changes to next value. ("-": Not linked, 1-9: Linked, "H": Hyperlink)
- (Link views)	Middle click		Opens a list of available links. Click any to select.
1 (Label)	Left click / Right click		Changes to previous/next channel that is on this label.

In Lissajous Views

Table D-2 Mouse and keyboard functions

Area	Action	Pointer	Function
(Channel list)	Left click		Displays a list of channels. Click any to select.
Channel name	Left click		Changes to previous channel in list.
Channel name	Right click		Changes to next channel in list.
(Data type)	Left click		Display a list to choose the data type.
Scale value	Left click		Change to previous scale in list.
Scale value	Right click		Change to next scale in list.
Scale value	Middle click		Opens a list of voltage values. Click any to select.
Scale value	Left click + vertical drag		Increases/Decreases the voltage value.
Signal area	Left click + drag		Moves the data representation inside the Lissajous.
Signal area	Left click		Centers data in the Lissajous.
Signal area	Hold CTRL + left click + drag		Rotates data in the Lissajous.
Signal area	Right click + drag		Changes amplitude scale of both axes.
Signal area	Right click		Opens a contextual menu.
Inside rectangle alarm zone	Left click + drag		Moves the alarm zone.
On border of rectangle alarm zone	Left click + drag		Increases/decreases the size of the zone.
Inside pie alarm zone	Left click + drag		Rotates the alarm zone.
On border of pie alarm zone	Left click + drag		Increases/decreases the size of the zone.

Table D-2 Mouse and keyboard functions *(continued)*

Area	Action	Pointer	Function
	Left click		Displays measurement values in the expanded strip chart title bar. It activates the title bar of the expanded strip chart.
0	Left click		Toggles between 0° and 180°.
+ DEF1	Left click		Adds the indication to the analysis report.
Arrow to the right	Left click		Opens and closes a list of indication codes.
Signal trace	Left click + drag		Move the signal trace up and down inside the expanded strip chart.
Blue cursors	Left click + drag		Increases or decreases the area covered inside the cursors.
Signal trace	Right click		Move the blue cursors to the mouse position.
Anywhere	Space bar		Center data in view.
100 (Frequency)	Left click / Right click		Changes to previous/next frequency for the same coil.
1 (Coil)	Left click / Right click		Changes to previous/next coil in the same frequency.
- (Link views)	Left click		Changes to previous value. (“-”: Not linked, 1-9: Linked, “H”: Hyperlink)
- (Link views)	Right click		Changes to next value. (“-”: Not linked, 1-9: Linked, “H”: Hyperlink)
- (Link views)	Middle click		Opens a list of available links. Click any to select.
0° (rotation)	Left click		Reduce rotation by 1°.
0° (rotation)	Right click		Increases rotation by 1°.
0° (rotation)	Middle click		Shows “Data History” dialog.
0° (rotation)	Left click + vertical drag		Changes rotation.

In Lissajous Views

Table D-2 Mouse and keyboard functions *(continued)*

Area	Action	Pointer	Function
1 (Label)	Left click / Right click		Changes to previous / next channel that is on this label.
Vpp; Vi; Vr; Vpf	Left click / Right click		Toggles between measurement modes: Vpp for peak to peak, Vi for vertical maximum, Vr for horizontal maximum and Vpf for peak first (peak to peak first transition).
°pp; °mr; °pr	Left click / Right click		Toggles between measurement modes: °pp for peak to peak, °mr for maximum rate, °pf for horizontal maximum and Vpf for peak first (peak to peak first transition).

In C-scan and Polar Views

Table D-3 Mouse and keyboard functions

Area	Action	Pointer	Function
	Left click		Opens and closes the list of C-scans. Click any to select.
Channel name	Left click		Changes to previous C-scan in list.
Channel name	Right click		Changes to next C-scan in list.
	Left click		Opens and closes the palette below the title bar.
	Left click		Opens and closes the navigation panel.
	Left click		Toggles the view among the horizontal, vertical, amplitude, and phase components.
Signal area	Right click + drag		Zooms in the image (C-scan only).
Signal area	Right click		Opens a contextual menu.
Signal area	Left click		Moves the cross selection cursor to that location.
Anywhere	Left click + drag		Moves the cross selection cursor.
Signal area	Left click + drag on cross extremities		Increases or decreases the area covered inside the cross selection cursor.
Signal area	Right click cross lines		Opens a contextual menu.
On the subtraction cursor	Left click + drag		Moves the subtraction cursor up and down.
Signal area	Middle click		Moves the subtraction cursor to that location.
On the subtraction cursor	Right click		Opens the Link contextual menu. Click the Link command to attach the subtraction cursor to the cross selection cursor.
100 (Frequency)	Left click / Right click		Changes to previous/next frequency for the same coil.

Table D-3 Mouse and keyboard functions *(continued)*

Area	Action	Pointer	Function
1 (Coil)	Left click / Right click		Changes to previous/next coil in the same frequency.
- (Link views)	Left click		Changes to previous value. (“-”: Not linked, 1-9: Linked, “H”: Hyperlink)
- (Link views)	Right click		Changes to next value. (“-”: Not linked, 1-9: Linked, “H”: Hyperlink)
- (Link views)	Middle click		Opens a list of available links. Click any to select.
1 (Label)	Left click / Right click		Changes to previous/next channel that is on this label.
Previous/Next Indication	Left click		Move to previous / next indication.

In 3D C-scan and 3D Polar Views

Table D-4 Mouse and keyboard functions

Area	Action	Pointer	Function
	Left click		Opens and closes the list of C-scans. Click any to select.
Channel name	Left click		Changes to previous C-scan in list.
Channel name	Right click		Changes to next C-scan in list.
	Left click		Opens and closes the dialog box for 3D C-scan properties.
	Left click		Opens and closes the palette on the right of the 3D C-scan view.
	Left click		Toggles the view between horizontal, vertical, amplitude, and phase components.
	Left click		Vertically flips data representation.
	Left click		Horizontally flips data representation.
Signal area	Right click + drag		Zooms data representation in and out.
Signal area	Left click + drag		Move the frames (in 3D polar view only).
Signal area	CTRL + left click + drag		Rotates the 3D representation on all axes.
Signal area	Right click		Opens a contextual menu (in 3D polar view only).
100 (Frequency)	Left click / Right click		Changes to previous/next frequency for the same coil.
1 (Coil)	Left click / Right click		Changes to previous/next coil in the same frequency.
- (Link views)	Left click		Changes to previous value. (“-”: Not linked, 1-9: Linked, “H”: Hyperlink)

Table D-4 Mouse and keyboard functions *(continued)*

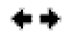
Area	Action	Pointer	Function
(Link views)	Right click		Changes to next value. ("-": Not linked, 1-9: Linked, "H": Hyperlink)
- (Link views)	Middle click		Opens a list of available links. Click any to select.
1 (Label)	Left click / Right click		Changes to previous/next channel that is on this label.
	Left click		Increases/decreases the X/Y ratio, the zoom and the pitch value.
Front	Left click		Changes the rotation configuration.
Color	Left click		Changes the surface color configuration.
	Left click		Increases/Decreases the rotation, the surface grid and the view.

In Side Views

Table D-5 Mouse and keyboard functions

Area	Action	Pointer	Function
or **X** or **Y**	Left click		Displays a list of channels or C-scans. Click any to select.
Channel name	Left click		Changes to previous channel in list.
Channel name	Right click		Changes to next channel in list.
	Left click		Toggles the view between horizontal and vertical, amplitude.
(Data type)	Left click		Displays a list to choose the data type.
100 (Frequency)	Left click / Right click		Changes to previous/next frequency for the same coil.
1 (Coil)	Left click / Right click		Changes to previous/next coil in the same frequency.
- (Link views)	Left click		Changes previous value. (“-”: Not linked, 1-9: Linked, “H”: Hyperlink)
- (Link views)	Right click		Changes to next value. (“-”: Not linked, 1-9: Linked, “H”: Hyperlink)
- (Link views)	Middle click		Opens a list of available links. Click any to select.
1 (Label)	Left click / Right click		Changes to previous/next channel that is on this label.
Scale value	Left click		Changes to previous voltage value.
Scale value	Right click		Changes to next voltage value.
Scale value	Middle click		Opens a list of voltage values. Click any to select.
Scale value	Left click + vertical drag		Increases/Decreases the voltage value.
Measurement cursor	Left click + horizontal drag		Moves the cursor left and right.

Table D-5 Mouse and keyboard functions *(continued)*

Area	Action	Pointer	Function
Measurement cursor	Double left-click		Moves the cursor to the intercepting closest data point crop line.
Crop Line	Left click + vertical drag		Moves the line up and down.
Signal Area	Right click + drag up or down		Increases or decreases the voltage value.
Signal Area	Left click + drag up/down		Moves data representation up and down inside the view.
Signal Area	Left click + drag left/right		Moves the signal left and right
Signal Area	Right click		Contextual menu.

General Keyboard Shortcuts

Table D-6 Keyboard shortcuts

Where	Area	Action	Pointer	Function
Anywhere		ALT + F4		Closes Magnifi.
Anywhere		CTRL + P		Opens the print screen dialog box.
Anywhere		F1		Opens the User's manual.
Anywhere		Space bar		Centers data in the different opened views.
When an instrument is connected		F2		Starts/Stops an acquisition.
When an instrument is connected		F4		Starts acquisition of the next file.
When an instrument is connected		F6		Balances probe(s).
When acquiring		F5		
When some trigrams displayed in code window		F7		Goes to the previous trigram.
When some trigrams displayed in code window		F8		
Inspection mode		F10		Starts/Stops saving data.
Acquisition mode		Delete		

Miscellaneous

Table D-7 Mouse and keyboard functions

Where	Area	Action	Pointer	Function
Code window		Left click / Right click / Middle click		Toggle between channel code window and C-scan code window.
Code window	indication area	Right click		Opens a contextual menu.
Analysis mode		Left click Next button		Load next data file.
Analysis mode		Right click Next button		
Acquisition mode	File list	Right click Row + x		Add a new tube, with Row less x.
Acquisition mode	File list	Left click Row + x		
Acquisition mode	File list	Right click Col + x		Add a new tube, with Col less X.
Acquisition mode	File list	Left click Col + x		
Acquisition mode	File list	Left click Insert		Add a new tube.
Acquisition mode	File list	Right click Insert		
Acquisition mode		Delete		Delete selected tube.

T +1 418-780-1565
F +1 418-780-2354
info@eddyfi.com

www.eddyfi.com

2013-11-04